APPENDIX D - COMPUTER PROGRAMS

TABLE OF CONTENTS

TABLE OF CONTENTS, (Cont'd)

APPENDIX D - COMPUTER PROGRAMS

LIST OF TABLES

LIST OF TABLES (Cont'd)

APPENDIX D - COMPUTER PROGRAMS

LIST OF FIGURES

LIST OF FIGURES (Cont'd)

LIST OF FIGURES (Cont'd)

APPENDIX D

COMPUTER PROGRAMS

The computer programs referred to in Sections 2.5, 3.7, 3.8, 3.9, and 3.10, Attachment C.A of Appendix C, by their acronyms are described herein. All programs are verified within the stated assumptions and limitations, for correctness of utilized theory and validity of obtained results for a variety of typical problems. Results are checked against known solutions, solutions obtained from other programs, or hand calculations. Examples of validation problems are included with the program descriptions. Whenever applicable, internal checks such as equilibrium and orthogonality checks are included as an aid in checking the validity of the results.

D.1 CBEAM

CBEAM (Reinforced Concrete Beam Design and Schedule) is written to perform the routine work of reinforcement selection for rectangular cross section beams. The program is based on the design methods of the ACI 318-71 Code and Sargent & Lundy's structural design standards.

In CBEAM, all beam sections are assumed to be rectangular sections. For stirrup reinforcement, each beam is divided into three portions: left 1/4 length, middle 1/2 length, and right 1/4 length. The program assumes that constant shear forces are applied within each region. Design forces (bending moments and shear forces) for continuous frames should be obtained from analysis programs such as STRUDL. Design forces for individual members should be obtained by any acceptable analytical procedure.

Required input data includes identification titles, dimensions of the member sections and design member forces. Output includes a beam schedule suitable for direct release for construction use and a longitudinal bar schedule.

To demonstrate the validity of CBEAM, a typical three span beam design was processed on CBEAM and the results compared to hand calculations.

Tables D-1 through D-3 show the beam characteristics and the resulting output for the three beams. As shown, the results compare very favorably.

D.2 CONCRETE

CONCRETE is a computer program used for statistical evaluation of concrete strength. It sorts and analyzes the field data collected on concrete samples and presents it in a convenient form for interpretation.

CONCRETE was developed and is maintained by Sargent & Lundy. Since 1972 the program has been used at Sargent & Lundy on UNIVAC 1100 hardware operating under EXEC 8.

The compressive strength test results of concrete cylinders are statistically analyzed to obtain the mean, standard deviation, coefficient of variation, moving averages, and other statistical parameters required in the quality appraisal of concrete according to ACI 214-65. The strength results are also compared with the quality control limits fixed according to the ACI 318-71 Code and the ASTM Manual on Quality Control of Materials, 15-C. Any violations or inadequacies are clearly pointed out in the output.

To demonstrate the validity of the program, a sample problem from "Notes on ACI 318-71 Building Code Requirements with Design Applications" (Reference 1) was processed on CONCRETE. The problem determines the average 28-day strength and standard deviation for 46 test cylinders. CONCRETE's results, shown in Table D-4, are identical to the hand calculation solutions in the ACI notes.

D.3 DYNAS

DYNAS (Dynamic Analysis of Structures) is designed to perform dynamic analysis of structures which can be idealized as three-dimensional space frames and/or rigid slabs connected together by translational or torsional springs. The program considers the combined effects of translational, torsional, and rocking motions on the structure. The program uses either the response spectrum or the time history method of analysis, depending on the type of forcing function available. Each method uses a normal mode approach. In the case of time history analysis, the decoupled differential equations of motion are numerically integrated using Newmark's β-method (Reference 3).

The DYNAS program is capable of analyzing structures having parts with different associated dampings. An option is also available to analyze a large structural system using a modal synthesis technique. In this option, the system is divided into subsystems whose modal characteristics are computed separately and then synthesized to obtain the response of the complete system. The input base motion can be applied simultaneously in two orthogonal directions. A response spectrum can be generated at specified slabs or joints.

The program output includes modal responses, probable maximum responses, a time history of structural response, and a response spectrum at specified joints.

The DYNAS program was originally developed by Sargent & Lundy in 1970. The program is currently maintained on UNIVAC 1100 series hardware operating under EXEC 8.

The solutions of two of the problems used for validating DYNAS are presented.

In the first problem, a three-story shear building is analyzed and compared to a solution obtained by Biggs (Reference 4). The structure in conjunction with the applicable masses and stiffness values is represented by the closed-coupled system shown in Figure D-1. For this analysis the following response spectrum was used:

Frequency	Displacement
1.00 cps	3.30 in.
2.18 cps	1.40 in.
3.18 cps	0.66 in.

The results obtained by Biggs and from DYNAS are compared in Tables D-5 through D-8.

In the second example, results of DYNAS are compared to those obtained by Wilson et al. (Reference 2) using the SAP-IV program.

An acceleration is applied at the fixed end of a cantilever beam (Figure D-2). The natural periods calculated by both SAP-IV and DYNAS are shown in Table D-9. A comparison of the bending moment at the fixed end of the cantilever beam is shown in Figure D-3.

As demonstrated in both examples, DYNAS performs an accurate analysis.

D.4 DYNAX

DYNAX (Dynamic Analysis of Axisymmetric Structures) is a finite element program capable of performing both static and dynamic analyses of axisymmetric structures. Its formulation is based on the theory of small displacement.

Three types of finite elements are available: quadrilateral, triangular, and shell. The geometry of the structure can be general as long as it is axisymmetric. Both isotropic and orthotropic elastic material properties can be modeled. Discrete and distributed springs can be used to model elastic foundations, etc.

For static analysis, input loads can be structural weight, nodal forces, nodal displacements, distributed loads, or thermal loads. Loads can be axisymmetric or nonaxisymmetric. For solids of revolution, the program outputs nodal displacements and element stresses in the global system (radial, circumferential, and axial) and element and nodal stress resultants in a shell coordinate system. For shells of revolution, the output consists of nodal displacements as well as element and nodal stress resultants in a shell coordinate system (meridional, circumferential, and normal).

For dynamic analysis, three methods are available: direct integration method, modal superposition method, and response spectrum method. Dynamic analysis by direct integration or modal superposition method uses a forcing function input via either 1) nodal force components versus time for any number of nodes, or 2) vertical or horizontal ground acceleration versus time. For nonaxisymmetric loads, the equivalent Fourier expansion is used. Dynamic analysis by the response spectrum method uses a spectral velocity versus natural frequency input with up to four damping constants. The output of dynamic analysis provides nodal displacements, element stresses, and resultant forces and moments at specified time steps. When the modal superposition method is used for earthquake response analysis, the prescribed number of frequencies and mode shapes are computed and printed along with the cumulative response of all specified modes by the root sum square (RSS) method and the absolute sum method.

DYNAX was originally developed under the acronym ASHAD by S. Ghosh and E. L. Wilson of the University of California, Berkeley, in 1969 (Reference 5). It was acquired by Sargent & Lundy in 1972 and is maintained on UNIVAC 1100 series hardware operating under EXEC 8.

Validation of the major analytical capabilities of DYNAX is demonstrated by a comparison of the results from six documented problems with DYNAX results.

The first problem is taken from S. Timoshenko and S. Woinowsky-Krieger's book Theory of Plates and Shells (Reference 6). A clamped shallow spherical shell, shown in Figure D-4, is analyzed for displacements and stresses produced by a uniform pressure applied on its outside surface. DYNAX and the Timoshenko/Woinowski-Krieger solutions are compared in Figures D-5 and D-6.

The second problem, taken from Theory of Elasticity by Timoshenko and Goodier (Reference 7), is a plane strain analysis of a thick-walled cylinder subjected to external pressure. The finite element idealization and the loading system used for this case are shown in Figure D-7. Results of the DYNAX analysis are compared with the exact solution in Figure D-8. The agreement for both stresses and displacements is excellent.

The third problem is taken from an article by Budiansky and Radkowski (Reference 8). The structure, illustrated in Figure D-9, is a short, wide cylinder with a moderate thickness to radius ratio. The applied loads and the output stresses are pure uncoupled harmonics. For this finite element analysis, the cylinder is divided into 50 elements of equal size. This problem solves for harmonic deflections, element stresses, and forces. Figures D-10 and D-11 compare DYNAX results with the results given in the article.

The fourth problem is taken from an article by Reismann and Padlog (Reference 9). A ring (line) load of magnitude P (500 pounds) is suddenly applied to the center of a freely supported cylindrical shell. The dimensions of the shell and the time-history of the load are shown in Figure D-12. Because of symmetry, only one-half of the cylinder is modeled using 80 elements of equal size. The time-history of radial deflection and meridional moments from DYNAX and from Reismann and Padlog are compared and are shown in Figures D-13 and D-14, respectively.

For the fifth problem, the method of mode superposition is used to solve a shallow spherical cap with clamped support under the action of a suddenly applied uniformly distributed load. The dimensions of the shell and the load time-history are shown in Figure D-15. The first 12 modes were considered to formulate the uncoupled equations of motion. Each of these equations was solved by the step-by-step integration method using a time step of 0.1×10^{-4} seconds. The results are compared graphically with those obtained by S. Klein (Reference 10) in Figures D-16 and D-17.

The sixth problem is a hyperbolic cooling tower, as shown in Figure D-18. The tower is analyzed for horizontal earthquake motion. A response spectrum for 2% damping, as shown in Figure

D-19, was used for this analysis. The RMS values of the meridional force are compared with those obtained by Abel et al. (Reference 11) in Figure D-20.

As shown in these six examples, DYNAX is capable of producing accurate results for both static and dynamic analyses of shells.

D.5 LAFD

LAFD (Analysis of Linear Anchor Forces and Displacements) calculates the maximum force and displacement of anchors resulting from local buckling of thin plate liners anchored to concrete walls. The solution method used in LAFD is described in Reference 12.

First, anchor displacements are found for an assumed postbuckling load by a relaxation technique. Then, using this maximum displacement, the anchor force and the strain in the buckled plate are calculated. The stress-strain relation given in a paper by Young and Tate (Reference 13) is reestablished in the program. Using the calculated strain, first stress is found and then a new load. The new load is then used to find a new set of displacements. The procedure is repeated to find a second new load. This load is then compared to the load used in the previous cycle. The procedure is repeated until the difference between the loads obtained in the last two cycles is approximately zero.

The program is capable of analyzing four types of anchors: Nelson studs of 1/2-, 5/8-, and 3/4-inch diameter, and 3- x 3-1/4-inch angle continuous rib anchors. The force-deformation relations of these anchors are obtained from the manufacturer's publication (Reference 14).

The program output includes the maximum anchor force, the maximum anchor deformation, and the postbuckling load of the buckled plate.

LAFD was developed by Sargent & Lundy in 1971 and is currently maintained on UNIVAC 1100 series hardware operating under EXEC 8.

To validate the program, significant calculations were verified with hand calculations. As an example of this validation, a comparison of these calculations is presented for a strip of liner having the following properties:

Strip span a	=	17.5 in.,
Plate thickness t	=	0.375 in.,
Strip width w	=	9 in.,
Modulus of Elasticity E	=	30×10^3 ksi, and
Yield Stress β_o	=	36 ksi.

5/8-inch-diameter Nelson studs are used as anchors.

The anchor displacements, U_i, the force in the anchor adjacent to the buckled panel, f_1, and the postbuckling load P as calculated by the program are shown in Table D-10. Substituting these displacements into the appropriate force-deformation relationship for a 5/8-inch-diameter Nelson stud yields the anchor forces contained in Table D-11.

The validity of the solution is checked using the displacements and anchor forces given in Tables D-10 and D-11 for the system shown in Figure D-21 to verify the equality of the original equations:

$$Fo - P = \frac{EA}{a}(U_1 - U_2) + f_1 \qquad (1)$$

$$O = \frac{EA}{a}(2\,U_n - U_{n-1} - U_{n+1}) + f_n \qquad (2)$$

$$n = 1, 2, 3....N$$

The postbuckling load, P, as determined by Equation 1, is equal to 21.864K as compared to 21.978K obtained from the program. Substitution into Equation 2 satisfies the equation; equilibrium having been verified, the results obtained from the program are valid.

D.6 SIMULATE

SIMULATE (A Monte-Carlo Analysis of the Turbine Hazard) performs a Monte Carlo Simulation analysis of the turbine missile hazard to nuclear power plants. It models the plant structures, safety-related equipment and turbine disc characteristics. SIMULATE traces each missile's path through the plant structures, considering cases of perforation and ricochet, and recording damage to the safety-related equipment until the energy of the missile drops below a preselected level or the missile leaves the plant region.

The program also stores information on a system file, such as the trial number in which one or more targets are hit, the hit target names, and the corresponding missile weight. Such information may be used subsequently to determine frequencies of those combinations (of targets) which are detrimental to the safety of the plant.

SIMULATE was developed at Sargent & Lundy in 1977. It is currently maintained in UNIVAC 1100 series hardware operating under EXEC 8.

The input consists of information describing the plant structures and safety-related components as well as the postulated missile characteristics.

The output includes the information necessary to evaluate the probability of damage that would interfere with normal safe shutdown beyond certain limits.

To validate SIMULATE, five problems were used. The first two problems are for the validation of the modified NDRC formula and CEA-EDF formula for the perforation of concrete barriers, and the Ballistic Research Laboratory (BRL) formula for the perforation of steel barriers. The other three problems deal with the extraction of frequencies of a given set of combinations defining a damage state from a given set of targets which are hit in a given number of trials.

D.7 PCAUC

PCAUC (Portland Cement Association Ultimate Design of Columns) is used to design or to investigate reinforced concrete columns using the ultimate strength theory in accordance with ACI 318-71 Code. The program is capable of designing or investigating tied columns subjected to an axial load combined with uniaxial or biaxial bending moment. The program input consists of the dimensions of sections, material properties, reinforcement requirement and loading data. The applied forces output includes the load factors per ACI. The slenderness effect is not included in the present program.

Output from the design part of the program includes the steel reinforcement arrangement, ultimate capacity for all loading cases, and interaction control points data. Output from the investigation part of the program either includes biaxial or uniaxial interaction data. Sargent & Lundy has modified the original PCA program to follow 1971 ACI building code and to provide more design options and greater capacity.

PCAUC is a modified version of the program "Ultimate Strength Design of Concrete Columns", developed by the Portland Cement Association. The program was obtained by Sargent & Lundy in 1972 and modified. It is currently maintained on UNIVAC 1100 series hardware operating under EXEC 8.

To validate PCAUC, documented results from several problems were compared with PCAUC results. Three of these problems are presented here.

The first problem is taken from Wang and Salmon's book (Reference 19). The reinforcement for a 17-inch x 17-inch square tied column is designed for compression control loads. The loads include a dead-load axial load of 214 kips and bending moment of 47 ft/kips, and a live-load axial load of 132 kips and a bending moment of 23 ft/kips. The reinforcement is designed according to the ACI Code with f_c' = 3,000 psi and f_y = 40,000 psi.

The solution as given in Wang and Salmon's book is identical to the solution obtained from PCAUC, shown in Figure D-22. It should be noted that the ultimate capacity provided by PCAUC has been reduced by a factor of 0.7.

The second problem is also taken from Reference 19. The reinforcement for a tied column 14 inches wide and 20 inches deep is designed for tension control loads with a dead-load axial load of 43 kips and bending moment of 96 kips, and a live-load axial load of 32 kips and bending moment of 85 ft/kips. The reinforcement is designed according to ACI Code using symmetrical reinforcement with respect to its width and with f_c' = 4,500 psi and f_y = 50,000 psi.

The solution as given in Wang and Salmon's book is identical to the solution obtained from PCAUC, shown in Figure D-23.

The third problem is taken from Notes on ACI 318-71 Building Code Requirements with Design Applications (Reference 20). A square tied column 28 inches x 28 inches is designed for biaxial bending loads for the following service loads:

	Dead	Live
Axial	550 kips	300 kips
M_x	320 ft/kips	200 ft/kips
M_y	160 ft/kips	100 ft/kips

The bending is designed according to the ACI Code with fc' = 5,000 psi and f_y = 60,000 psi.

The selected reinforcement obtained from PCAUC, shown in Figure D-24, is identical to that from Reference 20. It should also be noted that the interaction control points obtained by both show good agreement.

D.8.1 PIPING ANALYSIS PROGRAMS

PIPSYS (Integrated Piping Analysis System) analyzes piping systems of power plants for static and dynamic loadings, and computes the combined stresses. The following analyses are performed:

a. Static: Analysis of thermal, displacement, distributed, and concentrated weight loadings on piping systems;

b. Dynamic: Analysis of piping system response to seismic and fluid transient loads;

c. Stress Combination: Computes the combined stresses in the piping components in accordance with the ASME Boiler and Pressure Vessel Code, Section III (Reference 21).

The static, dynamic, and stress combination analyses can be performed independently or in sequence. Results of the static and dynamic analyses can be stored on magnetic tape for use at a later date to perform the stress combination analysis. The piping configuration can be plotted on a Calcomp plotter.

The input consists of the piping system geometry, material properties, static and dynamic loadings. Various options exist to control the length of the output. The default option generally prints only the summary of input data and final results.

PIPSYS was developed at Sargent & Lundy in 1972. It is currently maintained on UNIVAC 1100 series hardware operating under EXEC 8.

To demonstrate the validity of the PIPSYS program the following three examples are presented.

To illustrate the validity of the static portion of PIPSYS, the problem shown in Figure D-25 was analyzed and the results compared to those given in Reference 22. Table D-13 shows the comparison of member end moments. As shown, the results from PIPSYS and Reference 22 are in good agreement.

To illustrate the validity of the stress combination analysis portion of PIPSYS, the problem outlined in Reference 23 was reanalyzed on the PIPSYS program. The layout of the piping system is shown in Figure D-26. The stress analysis is performed at location 19. The summary of load sets and descriptions is presented in Table D-14. The results of the stress analysis are presented in Tables D-15 and D-16. The notations and equation numbers correspond to the ASME Boiler and Pressure Vessel Code (Reference 21).

It is observed that the PIPSYS results are very close to those presented in Reference 23.

To illustrate the validity of the dynamic analysis portion of PIPSYS, a problem was analyzed and the results obtained from PIPSYS were compared with those from two public domain computer programs, DYNAL (Reference 24) and NASTRAN (References 25 and 26).

Figure D-27 shows a schematic representation of the piping system analyzed. The system is modeled with simple beam elements with a total of 136 degrees of freedom. Figure D-28 shows the time dependent blow-down forces at the relief valves locations. Results of PIPSYS are compared with DYNAL and NASTRAN in Table D-17 and Figure D-29. The results from all three programs are quite close.

D.8.2 WESTDYN

The WESTDYN computer program is a Westinghouse proprietary code for the analysis of three-dimensional piping systems. WESTDYN performs linear, elastic analyses of piping systems subjected to internal pressure, static, thermal, and seismic loads. The program combines output loads in accordance with ASME Section III (Reference 21) or ANSI B31.1 piping stress criteria to arrive at actual piping stresses.

The piping system to be analyzed may contain a number of sections, a section being defined as a sequence of straight and/or curved members lying between two network points. A network point is: (a) a junction of two or more pipes; (b) an anchor or any point at which motion is prescribed; or (c) a position of lumped mass. A network point may be defined as completely unrestrained, or one or more of its six degrees of freedom may be rigidly or elastically constrained or displaced. Any member in the system may sustain prescribed loads. Also, at any location within the system members may be changed, masses concentrated, springs inserted, temperature conditions varied, materials and weld configurations changed, and body forces altered.

WESTDYN computes at each point within the piping system the forces, moments, translations, and rotations which result from the imposed anchor or junction loads, thermal gradients in the system, and gravitational loads in any combination of the three orthogonal axes. For seismic effects, a normal mode analysis is performed using three-dimensional response spectra. The resultant internal forces and moments are computed from the square root of the sum of the squares of the modal forces and moments.

D.8.3 CPASYS

CPASYS (Conversational Piping Analysis System) is a comprehensive system of interactive computer programs that were designed to automate and simplify piping design calculations. Pipe geometry and loading condition descriptions are permanently stored on project unique data base files. The control system program will retrieve the information and allow the piping analyst to maintain it.

The interactive programs in the system allow an analyst to perform all operations necessary to analyze a piping system, review the design of its interfaces, design welded attachments, and document all work performed. Design control tool documents are also generated.

Origin of Program: Sargent & Lundy

D.8.4 SIPDA

SIPDA (Simplified Piping Dynamic Analysis) is used to seismically qualify small piping subsystems ≤2 inches in diameter. In compliance with the limiting allowable stress and deflections, it considers the effects of pressure, weight and seismic loadings to calculate the maximum allowable span lengths. SIPDA was validated by comparison of sample problem results with previously validated results.

Origin of Program: Sargent & Lundy

D.8.5 NOHEAT

NOHEAT (Nonlinear Heat Transfer Analysis) uses the finite-element method to calculate the temperature distribution in an axisymmetric solid which results from nonlinear heat transfer. Stresses resulting from linear thermal expansion are calculated for the applicable model and for certain appropriate sections. NOHEAT was validated by comparison of sample problem results to previously validated results and manual calculation results.

Origin of Program: University of California (Berkeley)

D.8.6 HYTRAN

HYTRAN (Hydraulic Transient Analysis) calculates pressures, velocities and force transients in a liquid filled piping network due to transients that are initiated by valve closure, pump trip, or by pressure changes at a piping terminal. HYTRAN was validated by comparison of sample problem results to results given in published documents.

Origin of Program: Sargent & Lundy

D.8.7 PWRRA

PWRRA (Pipe Whip Restraint Reaction Analysis) computes the response of the simple pipe-whip analysis models to an applied time dependent blowdown force. It provides the load data required for the pipe whip restraint and the support structure.

PWRRA was validated by comparison of sample problem results with available analytical results in published technical literature.

Origin of Program: Sargent & Lundy

D.8.8 RELVAD

RELVAD (Relief Valve Design Program) is used in the design of safety/relief valve assemblies. The program calculates fluid forces at valve discharge exit and vent stack inlet and exit, moments and stresses in the discharge elbow, discharge flange, valve inlet weld and branch connection to the run. RELVAD was validated by comparison of sample problem results with results of example problem in ANSI Code For Pressure Piping, Winter 1975 Addenda to Power Piping ANSI B31.1g-1976.

Origin of Program: Sargent & Lundy

D.8.9 PWUR

PWUR (Rupture Analysis for Unrestrained Pipes) calculates the effect of a pipe rupture on the surrounding area. This program calculates the steady-state thrust coefficient as a function of the resistance coefficient of the piping system for steam and saturated on subcooled water, calculation of the component of the blowdown force due to steady state and the duration of the wave force and, calculation of the area affected by jet impingement as a function of the resistance coefficient. PWUR was validated by comparison of sample problem results with results of manual calculations.

Origin of Program: Sargent & Lundy

D.8.10 SRVA

SRVA (Safety Relief Valve Blowdown Analysis) is a finite difference program for the analysis of transient flow in a relief valve line discharging into a suppression pool. Transient forces and the pressures at the water column and the valve outlet are calculated. SRVA was validated by comparison of sample problem results with published results and analytical problem solutions.

Origin of Program: Sargent & Lundy

D.8.11 NONLIN

NONLIN (Nonlinear Dynamic Analysis of 2-D Structures) determines the nonlinear response of a complex structural or piping system model. Various material properties, forces and other parameters are input to generate the response. NONLIN was validated by comparison of sample problem results with results from an existing validated program (DRAIN-2).

Origin of Program: Sargent & Lundy

D.8.12 RELAP 4/MOD 5

RELAP 4/MOD 5 was used to obtain fluid velocities, densities and pressures for each time step which was used to calculate blowdown forces. These force time histories were then input into the PWRRA program. RELAP was validated by comparison of results of sample problem contained in the program file with previously validated results.

Origin of Program: EG&G

D.8.13 AXTRAN

AXTRAN (Axial Temperature Transients in Welds) performs a thermal transient analysis and generates an axial temperature profile on the stagnant line. AXTRAN was validated by comparison of sample problem results with results from an existing validated program (NOHEAT).

Origin of Program: Sargent & Lundy

D.8.14 ADINA

ADINA (Automatic Dynamic Incremental Nonlinear Analysis) performs nonlinear and linear static and dynamic finite element analysis. ADINA was validated by running test problems contained in the ADINA User's Manual.

Origin of Program: Massachusetts Institute of Technology

D.8.15 ANCHOR

ANCHOR (Analysis of Intermediate Anchors on Piping Systems) performs the anchor load combination for ASME Code NC and NF Loads. ANCHOR was validated by comparison of sample problem results with results of manual calculations.

Origin of Program: Sargent & Lundy

Subsections D.8.16 and D.8.17 describe piping analysis programs used on the Braidwood project only.

D.8.16 QUICKPIPE

QUICKPIPE performs support optimization and rigorous computer analysis of 4 inch and under, Class 2/3 small bore piping. The QUICKPIPE verification includes approximately 40 test runs, selected to "exercise" every portion of the program. Each run's results were verified against parallel SUPERPIPE results.

SUPERPIPE is a computer program for the rigorous analysis and design checking of piping systems. It was developed by Impell Corporation in 1974.

SUPERPIPE was benchmarked by comparison with results published by the NRC in NUREG/CR-1677 for seven sample problems. The comparison was performed in accordance with the NRC request for additional verification of computer codes used for analysis of nuclear piping systems. The verification specification addressed the response spectrum method of dynamic analysis commonly used in seismic qualification of nuclear piping. The program has also been thoroughly tested and verified for a comprehensive set of sample problems, including extensive comparison with several publicly available programs and ASME benchmark problems.

QUICKPIPE utilizes the 1974 ASME Code up to and including the Summer 1975 Addenda. All verification analyses have been documented in accordance with established Impell Quality Assurance procedures.

D.8.17 AUTOHANG

AUTOHANG is an interactive graphics computer program which designs pipe supports. It selects and analyzes component hardware, performs frame qualifications, and produces final drawings.

INTERSUPPORT is a comprehensive computer program for the structural analysis and design checking of pipe support structures. It uses a finite element beam analysis to calculate stresses and displacements. The program also performs a Raleigh frequency evaluation.

INTERSUPPORT was developed by Impell Corporation in 1981 and has been verified against results of the hand calculations and several publicly available programs.

AUTOHANG reads the service level allowables from a plant-specific database. Faulted allowables may be preset to a specific value, or AUTOHANG can determine the correct factor to be applied to the normal allowables, as specified in the ASME code. AUTOHANG utilizes the 1974 ASME Code including the Summer 1975 Addenda and the AICS 7th Edition.

The results of AUTOHANG analyses, as well as the methodology of these analyses, has been extensively verified. Approximately 100 benchmark cases were run. The AUTOHANG results were compared to hand calculations.

D.8.18 MLT*MOMENT.MOX

MLT*MOMENT.MOX (Moment Range and Transient Conversion) calculates moment ranges between user supplied load set data (e.g., moments, load set ID, multiplication factors) using a method consistent with the method used in the Byron fatigue data. This was done so that one-to-one comparisons could be made between Braidwood and Byron moment ranges for each load set pair.

Origin of Program: Sargent & Lundy

D.8.19 MLT*MOMENT.TRAN

MLT*MOMENT.TRAN (Moment Range and Transient Conversion) calculates NB-3650 thermal transient stress quantities for each specified enveloped load set using Byron fatigue data temperature values (which are in degrees Fahrenheit) and other input parameters. The resulting thermal transient stress quantities are then used as input for Sargent & Lundy fatigue elevations on Braidwood.

Origin of Program: Sargent & Lundy

D.8.20 OPTPIPE

The OPTPIPE computer program, developed by NUTECH Engineers, is a special purpose program which performs linear elastic static and dynamic analysis of three-dimensional piping systems arbitrarily oriented in space. The program can perform static analyses for dead weight, internal pressure, thermal effects, support displacements and externally applied loads. Dynamic analyses can be performed for earthquake loading represented by either an acceleration response spectrum or a time history. For dynamic time history analyses, either the modal superposition or direct integration procedure can be used. In the response spectrum approach, different spectra at different supports may be provided. In the time history analysis approach, different acceleration time histories at different supports may be provided, and different damping for each mode of the system may be input. The program has the option of computing modal damping by considering different damping in each component of the piping system.

In static analysis, joint displacements, member forces, and support reactions are output for the complete system. For dynamic time history analysis, the time histories of these parameters and their maximums are obtained for selected nodes and members. For dynamic response spectrum analysis, maximum joint displacements, member forces, and support reactions are determined by a combination of each of these parameters for each mode and for each set of earthquake directions. The total response due to the different modes may be obtained by the absolute summation method, square root of sum of the squares method, or by the closely spaced modes summation procedure (ten percent method or grouping method).

The piping system may be composed of four different types of elements.

a. Three-dimensional straight and curved pipe elements.

b. Three-dimensional beam elements which can be used to model rigid hangers, valves, etc.

c. Boundary elements which can be used to model spring hangers and may also be used to determine support reactions.

d. Substructure stiffness input element.

For curved pipes and tee or branch connections, stiffness and stress modification effects are automatically taken into account. For curved and straight pipes, conventional effective stress magnitudes are computed.

OPTPIPE contains plotting capabilities for plotting the geometry of a structure. It also has the ability to perform stress checks for ASME Class 1, Class 2 and Class 3 piping, based on the requirements of the 1977 Edition of the ASME Code including Addenda through Summer 1979.

D.8.21 ANSYS

The ANSYS computer program is a large-scale, general purpose computer program for the solution of several classes of engineering problems. Analysis capabilities include static and dynamic; elastic, plastic, creep and swelling; buckling; small and large deflections; steady state and transient heat transfer, electrostatics, magnetostatics, and fluid flow.

The matrix displacement method of analysis based upon finite element idealization is employed throughout the program. The ANSYS program is capable of analyzing two- and three-dimensional frame structures, piping systems, two-dimensional plane and axisymmetric solids, three-dimensional solids, flat plates, axisymmetric and three-dimensional shells and nonlinear problems including interfaces and cables.

Loading on the structure may be forces, displacements, pressures, temperatures or response spectra. Loadings may be arbitrary functions of time for linear and nonlinear dynamic analyses. Loadings for heat transfer analyses include internal heat generation, convection and radiation boundaries, and specified temperatures or heat flows.

ANSYS is a proprietary engineering analysis computer program developed by Swanson Analysis Systems, Incorporated.

D.8.22 GAPPIPE

The GAPPIPE computer program is a general purpose piping analysis program developed by Robert L. Cloud & Associates, Inc., and sponsored by the Electric Power Research Institute. GAPPIPE performs both linear and nonlinear elastic analyses of three-dimensional piping systems subjected to thermal expansion, imposed displacements, internal pressure, externally applied loads, and seismic and fluid transient loads or motions. In addition, GAPPIPE contains a postprocessor capable of performing stress evaluation of piping components in accordance with the ASME Boiler and Pressure Vessel Code, Section III requirements.

GAPPIPE differs from other piping computer programs in that it has the capability to analyze piping systems containing gaps. GAPPIPE has two analysis methods to compute the dynamic responses of such systems. The first method is nonlinear time history analysis by modal superposition and pseudoforce representation of gap responses. This method is most suitable for the simulation of piping responses induced by fluid transient loads or excitations where the input cannot be easily or adequately characterized by response spectra.

For excitation defined by response spectra, GAPPIPE offers a second analysis method that uses the response spectrum analysis technique and the method of equivalent linearization to account for the nonlinear behavior of gaps. In this method, GAPPIPE can use either uniform enveloped response spectra or different spectra at different supports using the independent support motion technique.

The GAPPIPE element library contains the following types of elements.

a. Three-dimensional pipe elements (straight and curved segments),

b. Boundary elements which are used to model supports, anchors, gaps and springs,

c. Three-dimensional truss elements, and

d. Three-dimensional beam elements.

Using the truss and beam elements, complex structures and equipment can be modeled and coupled with the piping models. A GAPPIPE analysis model can be a combination of one or more of the above element types.

This program is NRC-approved, as stated in Reference 55.

D.9 PLFEM-II

PLFEM (Plate Finite Element Method) analyzes plane elastic bodies, plates, and shell structures by the stiffness matrix method. The program uses two finite elements, a rectangular element and a triangular element.

Elastic spring supports and/or an elastic foundation may be considered in the analysis. Orthotropic materials may also be considered in conjunction with the rectangular element. Pressure loads, concentrated forces, nodal displacements, and thermal loads may be considered in the analysis. All loading cases may be factored and/or combined in any manner.

The program output includes deflections and rotations of all joints and membrane stresses (normal, shearing, and principal) at the center of each element, the resultant moments (X, Y, twisting principal), and shears and reaction forces. An equilibrium check is made to determine the accuracy of the results.

PLFEM was developed and is maintained by Sargent & Lundy. It was originally developed on a UNIVAC 1108 in 1966. Since May 1972 it has been successfully operating on the UNIVAC 1100 series hardware operating under EXEC 8.

Three sample problems are presented to demonstrate the validity of PLFEM. Plots of the computer results obtained are compared with theoretical results and results by other methods.

The first problem is an analysis of a rectangular tank filled with water which was presented by Y. K. Cheung and J. D. Davies (Reference 27). The finite element used was presented by Zienkiewicz and Cheung in August 1964 (Reference 28). Experimental results agreed exactly with the finite element results except at a few isolated points where very small differences were noted. The PLFEM grid and loading for the tank problem are shown in Figure D-30. The grid used is the same size as the one used by Cheung and Davies. Moments in three regions of the tank are plotted along with the PLFEM results in Figures D-31 through D-33.

As a second example, a rectangular plate subjected to a uniform plane stress and having a circular hole in its center is analyzed. The grid used in the PLFEM analysis is shown in Figure D-34. Because of double symmetry, only one quarter of the plate is analyzed. Results obtained from the PLFEM analysis are plotted in Figure D-35 against the exact values as given by S. Timoshenko and J. Goodier in Reference 7.

As a final example, a square plate having a rectangular hole in its center is analyzed for the effect of a thermal gradient through the plate. The grid used in the PLFEM analysis is shown in Figure D-36. Only one quarter of the plate is analyzed because of the double symmetry. Moment values obtained by PLFEM are plotted for two regions of the plate in Figure D-37. For comparison, values of the moments obtained by an analysis based on the Hrennekoff framework analogy are also shown.

D.10 RSG

RSG (Response Spectrum Generator) generates dynamic response spectra (displacement, velocity, and acceleration) for single-degree-of-freedom elastic systems with various damping, subjected to a prescribed time-dependent acceleration. The differential equation of motion is solved using Newmark's method of numerical integration (Reference 3).

The program may also be used to obtain a response-spectrum-consistent time-history in which the response spectrum of the generated time-history closely envelops the given spectrum.

The program has the capability of plotting the input time, acceleration function, and the response spectra output on tripartite and/or acceleration versus period frequency grids.

Depending on the option, the program output includes the spectra of a given time-history or the response-spectrum consistent time-history.

RSG was developed by Sargent & Lundy in 1969. Since 1972 the program has been maintained on UNIVAC 1100 series hardware operating under EXEC 8.

One of the comparisons used for validation is presented.

The response spectrum for a one-degree-of-freedom damped system as presented by Biggs (Reference 4) was determined using RSG. The system was subjected to the sinusoidal ground acceleration shown in Figure D-38. A damping factor of 0.2 was used for this example. The response spectra obtained by Biggs and from RSG are also shown in Figure D-38. As demonstrated by this comparison, RSG generates an accurate response spectrum.

D.11 SEISHANG

SEISHANG (Seismic Analysis of Hangers) is used for the analysis and design of electrical cable and HVAC duct support systems. The program computes the allowable spans for cable trays and selects the proper member sections for various types of supports. The input load functions can be in the form of dead load, live load, or dynamic response spectra.

Program input consists of geometric data, material properties, member properties, and external loadings. Program output consists of allowable spans, member sizes, and mechanical response.

The allowable slenderness ratios used for design of compression members in HVAC and cable tray support designs are shown in Table D-45.

SEISHANG was developed at Sargent & Lundy in 1976. It is currently maintained on UNIVAC 1100 series hardware under EXEC 8.

To demonstrate the validity of the program, two problems are presented.

A typical cable tray, shown in Figure D-39, is analyzed and compared to the solution obtained by hand calculation. The results obtained from SEISHANG and by hand calculation are compared in Table D-18. The results show good agreement.

Two typical HVAC supports, shown in Figures D-40 and D-41 are analyzed and compared to the solution obtained from the DYNAS (see Subsection D.3). The results obtained from SEISHANG and from DYNAS are compared in Tables D-19 and D-20. The HVAC support shown in Figure D-40 is also analyzed with PlPSYS (see Subsection D.8.). The results obtained from SEISHANG and from PIPSYS are compared in Table D-21. The results show good agreement.

D.12 SHAKE

SHAKE (Soil Layer Properties and Response/Earthquake) is a program which computes response in a horizontally layered semi-infinite system subjected to vertically traveling shear waves. Strain-compatible soil properties are computed within the program. Earthquake motion can be specified at any level of the soil profile, and a resulting motion can be computed anywhere else in the profile. The method is based on the continuous solution of the shear wave equation. For soil liquefaction studies, plots of stress time-histories at various levels in a soil profile can also be obtained.

The input for the program includes property data for the soil profile, curves of strain versus shear moduli and damping ratios, and the input earthquake motion.

The output includes the strain-compatible soil properties, response spectra of object and computed motions, printer and CALCOMP plots of time-histories, Fourier spectra, and response spectra. Stress time-history plots are also included.

SHAKE originally was developed by John Lysmer and P. B. Schnabel of the University of California, Berkeley (Reference 29). It was modified by and is now maintained by Sargent & Lundy. It has been used on a UNIVAC 1100 series hardware operating under EXEC 8 at Sargent & Lundy since October 1972.

To verify Sargent & Lundy's version of SHAKE, results from the program were compared with results from a problem in a paper by Idriss and Seed (Reference 30).

The 100-foot layer of dense sand shown in Figure D-42 was analyzed. The properties of the sand were considered to be as follows:

Total unit weight = 125 pcf,

$(K_2)_{max}$ = 65, and

K_o = 0.5.

The parameter $(K_2)_{max}$ relates the maximum shear modulus, G_{max}, and effective mean pressure at any depth, y, below the surface as follows:

$$G_{max} = 1000\,(K_2)_{max}\,\sigma_m'^{\,1/2}$$

Where

$$\sigma_m' = \frac{(1 + 2K_o)}{3}\,\sigma_v'\ ,$$

K_o = coefficient of lateral pressure at rest, and

σ_v = effective vertical pressure at depth y.

Damping values and the variation of modulus values with strain were based on published data for sands (Reference 31).

The response of the sand layer was evaluated using the time-history of accelerations recorded at Taft during the 1952 Kern County earthquake as base excitation. The ordinates of this time-history were adjusted to provide a maximum acceleration of 0.15g.

The results obtained from SHAKE and the published results are compared in Figures D-43 and D-44. The maximum shear stresses and accelerations from both solutions are compared in Figure D-43; the response spectra of the surface motions are compared in Figure D-44. As illustrated in these figures, the two solutions compare favorably.

D.13 SLOPE

SLOPE (Slope Stability Analysis) utilizes the theory of equilibrium of forces to determine the factor of safety against sliding of any embankment or slope. It contains the Bishop, Fellenius, and Morgenstern-Price methods of two-dimensional stability analysis. In the Bishop and Fellenius methods, the factor of safety against failure is estimated along a circular surface of failure, whereas any arbitrary failure surface may be chosen for the Morgenstern-Price method.

The input includes the slope geometry, soil profile, soil properties (density, cohesion, and the friction angle) and the piezometric surface(s). The program also has the capability to introduce an earthquake loading assumed as a horizontal gravitational force. Once the problem is input, several options can be used to determine the factor of safety by the various methods. In addition, different stages such as end-of-construction, full-lake, and sudden-drawdown, can be considered in a single run.

The output includes factors of safety for each trial surface and a plot of the slope cross section having slope profile, soil profile, water table conditions, and failure surface for the minimum factor of safety.

SLOPE was developed and put under ICES (Integrated Civil Engineering Systems) by William A. Bailey at the Massachusetts Institute of Technology. It has been in the public domain since 1967. Sargent & Lundy currently uses the SLOPE version maintained by the McDonnell Douglas Automation Company on IBM 370 Series hardware (Reference 32).

D.14 SLSAP1

SLSAP1 (Sargent & Lundy Structural Analysis Program) performs static analysis for structures consisting of any of the following element types: three-dimensional truss, three-dimensional beam, plane stress or plane strain, two-dimensional axisymmetric solid, three-dimensional solid, thin shell and boundary. The stiffnesses of the elements are evaluated for linear elastic isotropic or orthotropic materials. The structural stiffness is obtained by assembling all the individual element stiffnesses. In static analysis each load case may include element loadings - thermal loads, pressure loads, gravity loads, and concentrated nodal loads. The program calculates the nodal displacements and forces or stresses in elements for multiple load cases.

The original version of the program, SAP, was developed by E. L. Wilson of the University of California at Berkeley and released in September 1970 (Reference 33). In 1973 Sargent & Lundy modified the program to enable it to analyze a mat on a nonlinear elastic foundation, with zero foundation stiffness in regions of mat uplift. The regions of zero stiffness represent the fact that the soil foundation can not carry tension stresses. The program operates on the UNIVAC 1100 series hardware operating under EXEC 8.

To show the validity of the program, a circular plate on a rigid no-tension foundation, as shown in Figure D-45, is analyzed. The program results are compared with results obtained Timoshenko and Woinowski-Krieger's method of solution (Reference 35). As shown in Figure D-46, the results are in excellent agreement.

D.15.1 SLSAP-IV

SLSAPIV (Sargent & Lundy Structural Analysis Program) performs static and dynamic structural analyses. The structure may consist of any of the following element types: three-dimensional truss, three-dimensional beam, three-dimensional solid, plane stress or plane strain, two-dimensional axisymmetric solid thick shell, thin shell, isoparametric shell, boundary spring or pipe. The stiffnesses of the elements are evaluated for linear elastic isotropic or orthotropic materials. The structural stiffness is obtained by assembling all the individual element stiffnesses. In static analysis each load case may include element loadings - thermal loads, pressure loads, gravity loads and concentrated nodal loads. The program calculates the nodal displacements and forces or stresses in elements for multiple load cases. There are four options available in SLSAPIV dynamic analysis: frequency calculations only, frequency calculations followed by response history analysis, frequency calculations followed by response spectrum analysis, and response history analysis by direct integration. The program performs the solution for eigenvalue/vectors using either the determinant search algorithm or the subspace iteration algorithm depending on the size of the problem. The output for the time-history analysis and the response spectrum analysis includes displacement of the nodes and the element stresses.

The post processor, developed by Sargent & Lundy, enhances the working application of the static analysis portion of the SLSAPIV program. Its primary purpose is to perform load combination analyses for structures with multiple loading cases. The postprocessor combines files from independent runs into a single file, selects output requested by the user and checks for the absolute upper limits of the combined element stresses. It also has the capability to calculate the plate/shell minimum required moment capacities in two orthogonal directions or to calculate the principal stresses of the elements. In addition, computer graphic capabilities for contours have been implemented for the mat foundation.

SAP was originally developed by E. L. Wilson of the University of California at Berkeley in 1968. Sargent & Lundy currently maintains a modified SAPIV version released in 1973 (Reference 36) on UNIVAC 1100 series hardware operating under EXEC 8.

To demonstrate the validity of the major analytical capabilities of SLSAPIV, eight of the problems used for validation are presented. These problems are taken from Reference 36, which also contains comparisons with several other static and dynamic computer programs and classical solutions.

In the first problem, the pipe network shown in Figure D-47 is analyzed by SLSAPIV. The static response of the system is calculated under the combined effects of concentrated loads, vertical (y-direction) gravity loads, uniform temperature increase, and non-zero displacements imposed at one support point. The applied loads are shown in Table D-22.

The results from both programs are compared in Table D-23. Also shown are the results from Reference 37. As shown, all of the results compare favorably.

In the second problem, a clamped spherical shell shown in Figure D-48 is analyzed for stresses produced by a uniform pressure applied on its outside surface. The model represents a 5-degree wedge of the shell with eighteen thin-shell elements along the 39-degree meridian.

The curves in Figure D-48 are plots of the meridian (ϕ) and circumferential (θ) direction surface predicted by SAPIV (Reference 36) and SLSAPIV at the element centroid. The results are almost identical.

In the third problem a plane frame is analyzed to determine the three lowest frequencies and corresponding-mode shapes. The frame is shown in Figure D-49 (part a), and the beam element is shown in Figure D-49 (part b).

Results from Reference 36 and SLSAPIV are compared in Table D-24. As shown, the results compare favorably.

The fourth problem deals with the response spectrum analysis of a pipe assemblage. This problem was originally presented in Reference 38.

The model of the pipe assemblage is shown in Figure D-50. Z-moments are predicted for the local coordinates of the thirteen elements for the five lowest modes.

Table D-25 shows a comparison of the moment predictions from SLSAPIV and Reference 36. The proportional horizontal and vertical spectra are simultaneously specified. PIPDYN results, as documented in the SAPIV user manual, are also shown. All program results are in good agreement.

In the fifth problem a cantilever beam, shown in Figure D-51 (part a), is analyzed under both uniform and concentrated loads. The beam is modeled using 10 equal-length beam elements. It has a cross-sectional area of 1 x 2 inches, a length of 10 inches, and a Young's modulus equal to 30 x 10^3 ksi. A uniform load equal to 2 kips/inch and a concentrated load of 10 kips are applied at one end of the beam.

The results from SLSAPIV are compared to analytical results obtained by Timoshenko and Gere (Reference 34). Figure D-51 (part b) shows excellent agreement between the bending moments obtained by both solutions.

In the sixth problem a simply supported square plate under uniform loading is analyzed. A 10-inch-square by 1-inch-thick plate with Poisson's ratio equal to 0.3 and Young's modulus equal to 30 x 10^3 ksi is loaded with 1 ksi pressure.

The results obtained are compared to those presented by S. Timoshenko and S. Woinowski-Krieger (Reference 35). Bending moments M_{xx} and M_{yy} for both x and y symmetry lines obtained in the two solutions are shown in Figure D-52. The maximum bending moment which occurs at the center of the plate differs by only 1.05%.

In the seventh problem a cantilever beam, shown in Figure D-53 (part a) is analyzed for ground acceleration. The response history of eight flexural modes is calculated by mode superposition analysis. The ground acceleration applied at node 1 is shown in Figure D-53 (part b).

The natural periods for the eight lowest flexural modes as calculated by SLSAPIV and Reference 36 are given in Table D-26. The transverse deflection versus time for nodes 5 and 9 is plotted in Figure D-54. The fixed end moment versus time at element 1 is plotted in Figure D-55. The results show a favorable comparison.

For the eighth problem, the time history response of a cylindrical tube to a suddenly applied load is analyzed by mode superposition and direct integration. Results are compared with SAPIV and solutions by Reismann and Padlog (Reference 39).

One half of the tube, shown in Figure D-56 (part a) is idealized as an assemblage of axisymmetric elements with a total of 61 degrees-of-freedom. The time variation of the applied load is shown in Figure D-56 (part b).

The 20 lowest modes calculated by SLSAPIV and Reference 36 by mode superposition are listed in Table D-27. Figure D-57 shows the radial displacement versus time for SLSAPIV and Reference 39. Figure D-58 shows the plot for direct time integration results from SLSAPIV and Reference 39. As shown, results from SLSAPIV compare favorably with results from both SAPIV and Reference 39.

D.15.2 SAP90

SAP90 is a finite element program for the static and dynamic analyses of structural systems. The structural systems that can be analyzed on SAP90 may be modeled by one or a combination of the following element types:

a. three-dimensional frame (beam) element,

b. three-dimensional shell element,

c. two-dimensional solid element, and

d. three-dimensional solid element

The two-dimensional frame, truss, membrane, plate bending, axisymmetric, and plane strain elements are all available as special cases of the four elements named above. A boundary element in the form of translational or rotational spring supports is also available in the program. The type of loads allowed by the program include gravity, thermal, prestress, distributed, or nodal forces and prescribed displacements.

The program can perform static, steady-state, eigenvalue, and dynamic analyses. The static and dynamic analyses may be activated together in the same run, and load combinations may include results from both analyses. The dynamic analyses may include response spectrum or time-history analyses. In the time-history analyses, loading can be nodal load or base acceleration, and the solution is obtained using standard modal superposition or the Ritz vectors. The effect of an axial load on the transverse bending behavior of frame element can be considered by using the P-Delta analysis.

Two design postprocessors are available for frames analyzed by SAP90. SAPSTL uses the American Institute of Steel Construction Specifications (ASD-89 or PD-89 or LRFD-86) to check the design of steel frames analyzed by SAP90. SAPCON uses the American Concrete Institute Building Code Requirements for Reinforced Concrete (ACI 318-89) to design or check the design of concrete frames.

The input data consists of nodal coordinates, element members, loads, etc. The data are input to SAP90 in an unformatted file. The input file can be created interactively using SAPIN preprocessor program or noninteractively using a text editor. When SAPIN is used, the finite element model and input file for SAP90 analysis are generated graphically using pulldown menus to place joints and elements on the screen. Before executing SAP90, the user can use the interactive graphics program SAPLOT to plot the model on the screen, or on a hardcopy drawing. Important debugging options available through SAPLOT include display of joint restraints, element property ID, loading, and blowups of localized regions.

The output data consist of nodal displacements, element stresses or forces, and support reactions. All results are organized in output files stored on disk during the execution. Output data can be interactively plotted using SAPLOT and SAPTIME graphics programs. Important plotting options available through SAPLOT and SAPTIME include: deformed shape, nodal and element time history responses, contours of element stressed, principal and Von Mises stresses. Response spectrum curves for acceleration time history generated by SAP90 run can be generated and displayed using SAPSPEC.

SAP90 was developed by E. L. Wilson and A. Habibullah of Computers & Structures, Inc. (CSI), Berkeley, California.

D.16 SOR-III

SOR-III (Shell of Revolution) is a computer program used to analyze thin shells of revolution subjected to axisymmetric loading by employing a generalized Adams-Moulton method to integrate numerically the governing differential equations.

Arbitrary distribution of normal, tangential, and moment surface loadings as well as edge forces and deflections may be analyzed in the axisymmetric loadings. Input of boundary conditions allows for the consideration of elastic support conditions. Temperature variations along the meridian or across the thickness may also be considered.

The program output includes shell displacements, outer fiber stresses, and strains and stress resultants.

SOR-III was developed at Knolls Atomic Power Laboratory for the United States Atomic Energy Commission (Reference 40). Version III was acquired by Sargent & Lundy in 1969 and is currently maintained on Sargent & Lundy's UNIVAC 1100 series hardware operating under EXEC 8. The Sargent & Lundy version has been modified to punch data for plotting.

Results from this program have been frequently compared with other available solutions and other computer programs to check the validity of the program. One of these comparisons is the analysis of a circular flat reinforced concrete plate. The details of the problem and the boundary conditions are shown in Figure D-59. Results of the SOR-III analysis were compared with the finite element program, SABOR-III (Reference 41). Figure D-60 shows the bending moment in the meridional and hoop directions, respectively. Figure D-61 shows the comparison of radial shear. As shown in these figures, results compare favorably.

D.17 STAND

STAND (Structural Analysis and Design) is an integrated structural code which is programmed to perform analysis and design of structural steel members according to the 1969 AISC Specification. It consists of the following subsystems:

a. beam edit,

b. rolled beam design,

c. composite beam design,

d. plate girder design,

e. column edit,

f. column design, and

g. column base plate design.

The program input consists of member geometry and basic loadings. The design is performed for specified combinations of basic loadings and overstress factors. For floor framing systems, the program is capable of automatic transfer of reactions from tributary beams to supporting members. There are many design control parameters available, such as minimum and maximum depth limitations, shape of the rolled section, location of the lateral support of the compression flange, material grade or yield stress, deflection limitations, flange cutoff criterion, and location of stiffeners.

For columns, the program is capable of accounting for axial loading as well as uniaxial or biaxial bending.

For column base plate design, only axial load and column combinations are considered.

The program output includes the complete final design and provides the designer with sufficient intermediate information to enable him to evaluate the results. For rolled and composite beam designs, complete details of shop-welded and field-bolted end connections are contained in the output. Supplementary information for economic evaluation of the design is also provided.

STAND was developed and is maintained by Sargent & Lundy. Since May 1972, the program has been used extensively at Sargent & Lundy on UNIVAC 1100 series hardware operating under EXEC 8. Some of the principal applications include the design of steel floor framing using various types of horizontal structural elements and the design of columns or beam columns.

To validate STAND, results from the program were compared with results from example design problems in the Manual of Steel Construction (Reference 42). Four problems are given.

The first is a rolled beam design problem (Example 1, pp. 2-5). A beam of 36-ksi steel is designed for a 125-kip/ft bending moment, assuming its compression flange is braced at 6-foot intervals. The results, listed in Table D-28, show that STAND selects a more efficient section.

The second is a composite beam design problem (Example 1, pp. 2-143 and 2-144). A noncoverplated composite interior floor beam is designed. Limits of 1 1/2 inches for dead load deflection and 1 2/10 inches for live load deflection are imposed. The results, shown in Table D-29, are nearly identical.

The third is a column design problem with three examples, (Examples 1, 2 and 5, pp. 3-4, 5, and 9).

The first of these examples is the design of a W12 column of 36-ksi steel that will support a concentric load of 670 kips. The effective length with respect to its minor axis is 16 feet and to its major axis, 31 feet.

The second example is the design of an 11-foot-long W12 interior bay column of 36-ksi steel that will support a concentric load of 540 kips. The column, rigidly framed at the top by 30-foot-long W30 x 116 girders connected to each flange, is braced normal to its web at the top and the base.

The third example is the design of a W14 column of 36-ksi steel for a tier building of 18-foot story height that will support a 600-kip gravity load and a 190-kip/ft maximum wind moment, assuming K = 1 relative to both axes and bending is about the major axis.

The results from all three checks are identical to those in the AISC Manual, and are shown in Table D-30.

The fourth problem is a plate girder design problem (Example 1, p. 2-108). A welded plate girder is designed to support a uniform load of 3 kips/ft and two concentrated loads of 70 kips as shown in Figure D-62. The compression flange of the girder is supported laterally only at points of concentrated load. The results are shown in Table D-31.

D.18 STRUDL-II

STRUDL-II (Structural Design Language) is used primarily for static analysis of frame and truss structures. The program is, however, capable of performing linear static or dynamic analyses for finite element representations of structures using stiffness matrix methods. Nonlinear static problems and stability problems may also be treated.

The program is capable of analyzing plane trusses and frames, grids and elastic bodies, space trusses and frames, or three-dimensional elastic solids subjected to arbitrary loads, temperature changes, or specified displacements. Either earthquake accelerations or time-history force may be used for dynamic analysis. Anisotropic materials may also be used. In addition to analysis, the program is capable of performing structural steel design according to AISC Code and reinforced or prestressed concrete design according to ACI Code.

The program output depends upon the type of finite element used and the analysis that was performed. Included in the output are displacements and member forces and moments or element stresses and moments. Eigen values, eigen vectors, and time-history response or nodal response may be obtained for dynamic analyses. Member sizes may be obtained if the design portion is used.

STRUDL-II was developed as part of the Integrated Civil Engineering System at the Massachusetts Institute of Technology (Reference 43).

The program has been in the public domain since 1968. Two versions are currently being used, one maintained by the McDonnell Douglas Automation Company on IBM 370 series hardware (Reference 44) and one maintained by UNIVAC on the 1100 series hardware (Reference 45).

D.19 TEMCO

TEMCO (Reinforced Concrete Sections Under Eccentric Loads and Thermal Gradients) analyzes reinforced concrete sections subject to separate or combined action of eccentric loads and thermal gradients. The program can also analyze reinforced concrete sections subjected to axial force and biaxial bending. The effect of temperature is induced in the section by reactions created by the curvature restraint. No thermal gradient can be specified when analysis under axial force and biaxial bending is desired.

The analysis may be done assuming either a cracked or an uncracked section. Material properties can be assumed to be either linear or nonlinear. The program is capable of handling rectangular as well as nonrectangular sections.

The program input consists of section dimensions, areas and location of each layer of reinforcing steel, loads, load combinations, and material properties.

The curvature and axial strain corresponding to the given eccentric loads (axial load and bending moment) are determined by an iterative procedure. Thermal gradient is applied on the section by inducing reactions created by the curvature restraint, i.e., there is no curvature change due to a thermal gradient on the section. The axial expansion is assumed to be free after thermal gradient is applied. An iterative procedure is employed again for finding the final strain distribution such that equilibrium of internal and external loads is satisfied.

The program output consists of an echo print of the input, the combined loads, final location of neutral axis, final stresses in steel and concrete, and final internal forces. Similar intermediate results (before thermal gradient is applied) can also be output if desired.

The program can be used to analyze a wide variety of reinforced concrete beams and columns, slabs, and containment structures subject to various combinations of external loads and thermal gradients.

The program was developed by and is maintained by Sargent & Lundy. Since February 1972, the program has been extensively used at Sargent & Lundy on UNIVAC 1100 series hardware operating under EXEC 8.

To demonstrate the validity of TEMCO, program results are compared with hand calculated results. Four example problems are considered. The section and material properties for each problem are given in Table D-32, along with the applied external forces and thermal gradients. Those for the fourth problem are given in Table D-34.

The first problem considered involves a section with two layers of steel under the action of a compressive force applied at the centerline of the section, a bending moment, and a thermal gradient. A cracked analysis of the section is required assuming nonlinear material properties.

The second problem considered involves a section with two layers of steel under the action of a tensile force applied at the centerline of the section, a bending moment, and a thermal gradient. A cracked analysis of the section is required assuming nonlinear material properties.

The third problem considered involves a section with two layers of steel under the action of a tensile force applied at the centerline of the section, a bending moment, and a thermal gradient. A cracked analysis of the section is required assuming linear material properties.

The fourth problem involves a section with 10 reinforcing steel bars under the action of a tensile force and biaxial bending. A cracked analysis of the section is required assuming nonlinear material properties.

The hand-calculated solutions were obtained according to the following procedure:

a. Assume the location of neutral axis and the stress distribution to be the same as those given by the program under the given mechanical loading.

b. Compute the strain distribution under the given mechanical loading.

c. Compute the stress resultants by integration and using the proper stress-strain relationships.

d. Check for equilibrium with external mechanical loads.

e. If equilibrium is satisfied, compute the curvature imposed on the section by the given thermal gradient.

f. Compute the final curvature by subtracting the thermal curvature from the mechanical curvature.

g. Compute the new axial strain such that equilibrium is satisfied keeping the curvature constant.

h. Compute the final stress resultants by integration and using the proper stress-strain relationships.

i. Compute the thermal moment.

j. Check for equilibrium and compare program results with hand-calculated results.

Results obtained using this procedure together with those computed by TEMCO for all four problems are presented in Tables D-33 and D-35.

It is concluded that results given by the program agree very well with results obtained by hand calculations and that equilibrium between internal and external forces is satisfied for all three problems.

D.20 PENAN

PENAN handles the analysis of axially symmetric solids of revolution, which are composed of orthotropic materials with temperature-dependent properties, and which are subjected to asymmetric and time-dependent heating and loading.

Mainly, the structure of this program is a combination of two suitably modified, axisymmetric finite element programs. The two programs are:

a. NOHEAT (Nonlinear Heat Transfer Analysis Program) by I. Farhoomand and E. Wilson, and

b. ASAL (Finite Element Analysis-Axisymmetric Solids with Arbitrary Loads), Dunham & Nickell.

Outlined below is a brief description of the program's most significant features.

a. PENAN is designed to handle automatic finite element mesh generation and plotting for various penetration assembly configurations.

b. It has a built-in material property bank covering temperature-dependent mechanical and thermal properties (including fatigue design parameters) for all Section III materials.

c. It forms optimal load and load-range combinations for the various code-specified loading categories and generates Fourier series coefficients for all asymmetrically applied loads.

d. Through repeated use of the same stiffness matrix and unit-load-stresses, the program can carry out multiple stress evaluations.

   The program calculates the allowable stresses for all stress categories, makes the necessary stress comparisons, and generates the entire Penetration Assembly Stress Analysis Report.

D.21 SEEPAGE

SEEPAGE (Two-Dimensional Steady-State Seepage Analysis Program) is a finite element program developed for analyzing various types of two-dimensional steady seepage flows through non-homogeneous anisotropic porous media such as flow through an earth dike; flow into wells; and seepage losses through a bed of canals, lakes, etc.

The program is capable of computing the pressure, potential function, stream function values, velocities in two directions on a vertical plane, and discharge values through vertical section lines in the flow domain. It can also determine the position of the free surface line and plot the flow net.

Input for this program consists of the geometry of the flow domain, directional permeability coefficients, and available pressure heads on the boundaries. Output consists of nodal point pressures, potential values, stream function values, velocities in two directions in every element, and discharge through specified sections. For seepage problems involving free surface, additional input is required, including the initial trial free surface, number of iterations for free surface, free surface correction factor, and error tolerance.

SEEPAGE was originally developed by Robert L. Taylor of the University of California at Berkeley. It has been extensively modified by Sargent & Lundy since 1972. It is now maintained by Sargent & Lundy on UNIVAC 1100 series hardware under EXEC 8. In order to validate the SEEPAGE program, two validation runs were made and the results were compared with solutions obtained by other analytical methods.

Figure D-63 shows a plane flow problem along with the finite element representation used in the SEEPAGE computer run. The discharge per foot width of dam computed by SEEPAGE is 90.26×10^{-4} cfs; that obtained by using Dupuit's Theory (Reference 46) is 90.00×10^{-4} cfs. Comparison of free surfaces determined by using SEEPAGE and those using Kozeny's solution (Reference 47) is shown in Figure D-64. They are in close agreement.

For the second problem, groundwater flowing into a well is analyzed by SEEPAGE and hand calculations. The hand calculations are based on the well formula for steady radial flow in an unconfined aquifer as given in Reference 48. Figure D-65 shows the finite element mesh configuration and permeability coefficients. The discharge obtained from SEEPAGE is 0.6791 cfs; that from the hand calculations is 0.6567 cfs.

D.22 COLID

COLID (Column Interaction Diagram) calculates the axial load and bending moment capacities using allowable material and section properties for rectangular and circular (solid and hollow) reinforced concrete sections to be output as plotted interaction curves. The program uses the ACI 318-77 Building Code (Reference 49) and ASME (Reference 50) Code Factors. These factors may be replaced by user-defined values.

The axial load and bending moment capacities are generated by moving the neutral axis from the extreme compression fiber across the section and by checking the strain compatibility between the steel and the concrete. A complete interaction diagram is obtained for all sections for both compression and tension axial loads as well as positive and negative bending moments.

COLID was originally developed at Sargent & Lundy in 1973. It is now maintained on UNIVAC 1100 series hardware under EXEC 8.

To demonstrate the validity of the program, four rectangular and circular reinforced concrete sections are considered.

The first problem is a test of the Stress Factors option for a rectangular section. The sign convention and definition of parameters for rectangular sections are shown in Figure D-66. Design parameters and stress factors used for the problem are given in Table D-36. Results from the program compare favorably with hand calculations, as shown in Table D-37, for the locations on the interaction diagram shown in Figure D-67.

The second problem tests the Ultimate Capacity and ACI Ultimate Capacity options for rectangular sections. Table D-38 contains the design input parameters. COLID results compare favorably with hand calculations, as shown in Table D-39.

The third problem is a solid circular tied column used to test the ACI ultimate capacity option. Design parameters are given in Table D-40. As shown in Table D-41, the results obtained from COLID compare favorably with hand calculations.

The fourth problem is a hollow circular tied column used to test the Hollow Column option for ACI ultimate capacity. Design parameters are given in Table D-42. The hand-calculated results shown in Table D-43 compare favorably.

The final problem is an ultimate capacity rectangular section analysis to test metric units. Parameters in Problem 2 were converted to metric and the problem was reanalyzed. English results were converted using hand calculations and compared with the COLID metric results. As shown in Table D-44, the results compare favorably.

D.23 COMPONENT ANALYSIS PROGRAMS

D.23.1 THERST

THERST is a Westinghouse proprietary computer code that is a one-dimensional heat transfer code used to calculate the time history throughwall transient effects. The program is used to calculate the following on a time-history basis:

- Throughwall temperature distribution (printed values at 11 points through the wall)
- Average temperature
- Linear temperature distribution, ΔT_1
- Nonlinear temperature distribution, ΔT_2

using a finite difference solution technique for circular cross sections with an adiabatic outside surface. The temperature variation on the inside surface is specific as series of linear ramps. A convection film coefficient is applied as a "resistance" between the input temperature distribution and the inside surface. The time-history film coefficient can be specified directly either as a series of linear ramps or the time variation in velocity. The code will calculate the film coefficient, h. The following method is used to calculate the film coefficient:

$$h = h_{force} + h_{free} \ (\text{Btu/hr-ft}^2\text{-}^\circ\text{F})$$

where

h_{force} = forced convection film coefficient

h_{free} = free convection film coefficient

This method was chosen to give a continuous variation in film coefficient when velocity goes to zero; also, it is slightly conservative to include both forced and free convection coefficients. The forced convection film coefficient for flow inside a pipe is

$$h_{force} = 26.461 \frac{K}{D} \left(\frac{D\rho v}{\mu}\right)^{0.8} Pr^{0.4}$$

where

K = thermal conductivity (Btu/hr-ft-°F)

D = inside diameter (in.)

ρ = fluid density (lb/ft^3)

fluid velocity (ft/sec)

μ = viscosity (lb/hr-ft)

Pr = Prandtl number = $\frac{C_p \mu}{K}$

C_p = specific heat at constant pressure (Btu/lb-°F)

The free convection film coefficient for a vertical pipe is

$$h_{free} = 0.555 \frac{K}{D} \frac{g\beta \rho^2}{\mu^2} \Delta T \, D_i^3 \, Pr^{0.25}$$

where

g = gravitational constant

β = temperature coefficient of thermal expansion (1/°F)

D_i = inside diameter (ft)

ΔT = temperature difference between inside surface of pipe and fluid (°F)

THERST uses the temperature-dependent material properties (C_p, μ, K, etc.) in the calculation of the film coefficient.

The results of THERST are saved on TAPE19 for later input to MAXTRAN79. This tape contains the run title, the time variation in water temperature, average temperature, ΔT_1, and ΔT_2.

D.23.2 MAXTRAN79

The program MAXTRAN79 is a Westinghouse proprietary computer code that calculates the secondary and peak stress intensities as defined by ASME Section III NB-3650, on a time-history basis for only the thermal effects (ΔT_1, ΔT_2, T_a, and T_b). The stresses are calculated for one transient; that is, ranges are not considered. The secondary (S_s) and peak (S_p) stress intensity equations become:

$$S_s = C_3 \, E_{ab} \left| \alpha_a T_a - \alpha_b T_b \right|$$

$$S_p = K_3 \frac{E\alpha}{2(1 - \nu)} \Delta T_1 + K_3 \, C_3 \, E_{ab} \left| \alpha_a T_a - \alpha_b T_b \right| + \frac{E\alpha}{1 - \nu} \Delta T_2$$

where all terms are as defined in ASME Section III NB-3650 and $\nu = 0.3$ and $\alpha = \alpha_a$.

MAXTRAN79 finds the time(s) of maximum/minimum secondary stress and maximum/minimum peak stress to be used in the postulation of pipe break locations and the calculation of usage factors, respectively. The value of maximum peak stress is chosen at the time of maximum alternating stress, where

$$S_{alt} = [S_p]\ K_e/2.0$$

and where

S_n = an input primary-plus-secondary stress representing the range in pressure and moment stress plus the thermal transient stress $(T_a - T_b)$ from another transient

$$S_n = S_n^{'} + S_s$$

$S_n^{'}$ = secondary pressure and moment stress

$$K_e = -2.3 + 1.1\ \frac{S_n}{S_m} 1 \text{ and } 1.0 \leq K_e \leq 3.3$$

(austenitic stainless steel)

The input of MAXTRAN79 consists of cards describing a node number and types of members (tee, branch, straight run, weld, and the like) for which the Code calculates the stress indices. Also input are the tapes generated from the THERST program. One tape must be available for each cross section and transient being considered. If a point of thermal discontinuity is being analyzed (for instance, the weld at a valve), two tapes must be input. The tape containing the pipe cross-section data is used to obtain $(\Delta T_1$, ΔT_2, and T_a. The tape containing the valve cross-section data is used to obtain T_b.

The output of MAXTRAN79 consists of the following:

- An echo of the input
- A table giving the maximum and minimum secondary and peak stresses and the corresponding time, T_{water}, $(\Delta T_1$, ΔT_2, T_a, and T_b
- An output tape for FATCON input giving member data, $(\Delta T_1$, ΔT_2, $(T_a - T_b)$ and T_a for the condition yielding maximum and minimum secondary and peak stresses
- Plots of secondary peak and alternating stresses versus time

D.23.3 FATCON

Program FATCON is a Westinghouse proprietary computer code which is used to perform fatigue analysis in accordance with the ASME B and PV Code, Section III, Subsection NB-3600, 1977 Edition through and including the Summer 1979 Addenda.

The program input consists of a combination of card images and cataloged MAXTRAN thermal transient tapes. The card images identify member type, properties, fatigue cycle data, and other problem specific information. The tapes - consisting of up to 19 separate tapes per problem run - supply temperature transient data for use in evaluation of Code equations (10), (11), 13), and (14).

The program is divided into two distinct portions. The purpose of the first portion is to calculate the cumulative usage factor using the peak stress data from MAXTRAN. Included in this portion of the program are the specific transients by label and type, temperature data (input) associated with each transient, number of cycles for each transient, and the pressure, along with the stress indices for the member. The transient combinations are identified individually and all pertinent data are listed including cumulative usage factor. When equation (10) is exceeded, a message appears and equation (13) without moment is printed. The second portion, identified by the heading SECONDARY, is used to maximize equation (13) independently of the usage factor calculations and prints out equation (13) without moment for various transient combinations.

D.23.4 WECAN

The WECAN computer program is a Westinghouse proprietary computer program that can be used to solve a large variety of structural analysis problems. These problems can be one-, two-, or three-dimensional in nature. It has the capability to do static elastic and inelastic analyses, steady-state and transient heat condition analysis, steady-state hydraulic analysis, standard and reduced modal analysis, harmonic response analysis, and transient dynamic analysis.

The WECAN program is based on the finite element method of analysis. The analyst must model, or idealize, the structure in terms of discrete elements and apply loadings and boundary conditions to these elements. The stiffness (or conductivity) matrix for each element is assembled into a system of simultaneous linear equations for the entire structure. This set of equations is then solved by a variation of the Gaussian elimination method known as the wave front technique. This type of solution makes it possible to solve systems with a large number of degrees of freedom using a minimum amount of core storage. The maximum number of allowed degrees of freedom in the wave front depends on the amount of core available which, in turn, depends on the type of analysis being performed.

The library of finite elements includes spars, beams, pipes, plane elements, axisymmetric solids of revolution elements, three-dimensional solids, plates, plane and axisymmetric shells, three-dimensional shells, friction interface elements, springs, masses, dampers, heat conduction elements, hydraulic conducting elements, convection elements, and radiation elements.

WECAN is organized so that additional structural elements can be added with a minimum of effort. Input formats are similar for all elements and all types of analyses. Input data are used in the static analysis with only minor modifications.

The program is in a continual state of development. No version is made available until it has been checked and determined to be free of errors.

D.24 MISLODS

MISLODS (Probability of Turbine Missile Damage), pronounced "missile odds," calculates the probability of damage to power plant structures due to turbine missiles. This probability, P_4, can be expressed as

$$P_4 = P_1 P_2 P_3$$

where

P_1 = probability of missile ejection,

P_2 = probability of the missile striking selected barriers, given the missile ejection,

P_3 = probability that the missile will penetrate the barriers, given the missile ejection and strike.

P_1 is supplied as input. MISLODS calculates the values of P_2 and P_3, which are dependent on the missile characteristics, plant geometry, structural materials, and thicknesses. The methodology is based on references 51, 52, and 53.

MISLODS was developed by Sargent & Lundy in 1975. It is currently maintained on UNIVAC 1100 series hardware operating under EXEC 8.

To demonstrate the validity of MISLODS, the program's solution is compared to the problem presented by Semanderes on pages 7 through 10 of reference 54. In this problem, strike probabilities are determined for 3 types of targets and 4 types of missiles. The comparison between MISLODS and Semanderes' results is shown in Table D-12. The comparison is not exact, however, because of the differences in the two methods. Where Semanderes uses a Monte Carlo simulation, MISLODS uses a more definitive numerical integration procedure. However, as shown in Table D-12 the independent solutions compare favorably.

D.25 BSAP

BSAP (Bechtel Structural Analysis Program) is a general purpose, finite-element computer program for analysis of structural systems subject to static, dynamic, and thermal loads. The program incorporates an extensive library of beam, shell, and solid elements, such that virtually any type of structure can be represented. The Bechtel version of the BSAP is based upon and incorporates features of the SAP program developed at the University of California at Berkeley by Professor E. L. Wilson.

BSAP has been extensively used in the design and analysis of nuclear power plant structures since the mid-1970s. BSAP has been used to analyze the Byron and Braidwood Unit 1 containment structures to assess the effects of the temporary construction opening created to accommodate activities associated with the steam generator replacement project.

A thin quadrilateral and triangular shell element that has membrane and bending properties has been used to develop a three-dimensional, finite-element model of the containment structure. Each node of the shell element has five degrees of freedom (three translations and two rotations). The rotation about an axis normal to the plane of the element is not defined. Static loads which may be considered include nodal forces, distributed pressures, differential temperatures, and boundary movements. The static solution is obtained using Gaussian elimination technique or Crout elimination technique.

The validation process for BSAP consists of a number of problems designed to check the full range of available BSAP capabilities. The BSAP results are compared with the benchmark results derived from independent methods of solution that have been previously validated or are generally considered to be correct. Hand calculations employing well-established computation methods were used for some problems. For many problems, a benchmark solution was obtained by using an independently programmed public domain program.

D.26 BWSPAN/BWSCAN

BWSPAN is a large, finite-element program using beam elements for the analysis of structural systems. BWSPAN's library of elements include various pipe and structural elements. BWSPAN performs static, response spectrum, and time history analyses of structural systems. Stress analysis options include ASME Section III Class 1, 2, and 3; ANSI B31.7 and B31.1 for piping; and ASME NF and AISC for structural steel. Additional capabilities include nonlinear (gapped) static and dynamic analyses and thermal stratification analyses. BWSPAN is capable of evaluating thermal stratification. BWSCAN is a stress postprocessor of BWSPAN.

D.27 BWSPEC

BWSPEC reads and processes output files generated by BWSPAN and presents the results in a consistent, logical fashion that can be readily understood by equipment designers who require interface loading information.

D.28 RESPECT

RESPECT generates response spectra from time histories. It also generates time histories in structures and spectra for attachments.

D.29 ATHOSBWI

ATHOS is a three-dimensional, thermal-hydraulic code using the methods of computational fluid dynamics (CFD). ATHOS is comprised of two programs: the geometric preprocessor GPP module and the thermal-hydraulic program ATHOS module. Part of the output from the GPP module is used as input to the ATHOS module.

ATHOS3 Mod-01, developed by CFD Research Corporation (CFDRC) and licensed by EPRI, was acquired in March 1993 by B&W. B&W corrected some inconsistencies and made further modifications. The update was verified against field measurements and another CFD code, THIRST, which was developed by AECL. The program was renamed ATHOSBWI and has been used extensively at B&W for PWR replacement steam generator designs.

Major modifications to the original code ATHOS3 Mod-01 include corrections to the thermodynamic properties of the secondary fluid and the tube gap velocity calculations for flow-induced vibration (FIV) analysis. Since the latter is not used in the present study, only the thermodynamic property modification is described below. When ATHOS3 Mod-01 was obtained, it was found that the secondary saturation conditions were inconsistent with ASME steam tables. ATHOS3 Mod-01 correlations were then replaced with "IFC formulation for industrial use" from the ASME steam tables. These modifications were tested by comparisons with the saturation temperatures of the steam tables and with field measurements performed by Westinghouse.

The ATHOS3 code computes the steady-state and time-dependent behavior of the thermal-hydraulic parameters of PWR steam generators. The calculated overall (i.e., global) parameters of a steady-state analysis are:

a. Inlet temperature of the primary fluid,

b. Circulation ratio,

c. Secondary side inventory (i.e., liquid "collapse" level and associated liquid "hold-up" volume fraction) in the shell (excluding the downcomer region),

d. Enthalpy and flow rate of recirculating mixture, and

e. Downcomer liquid inventory, mass flow rates and average enthalpies.

In a transient analysis, the following overall parameters are calculated as a function of time:

a. Temperature drop of the primary fluid,

b. circulation ratio,

c. Shroud secondary side inventory (i.e., collapse level and hold-up volume fraction),

d. Enthalpy and flow rate of recirculating mixture,

e. Downcomer mass flow rates and average enthalpies,

f. Height of the water level in the downcomer, and

g. Rates of heat transfer from the primary side and to the secondary side; and either: (a) mass outflow rate of steam leaving the dome, or (b) steam dome pressure, depending on user-specified outlet boundary condition.

In order to calculate the above overall/global parameters, the ATHOS3 code first computes the three-dimensional distributions of the following parameters:

a. Primary-fluid temperature,

b. Tube-metal mid-wall temperature,

c. Heat flux to the secondary side fluid (steam and water mixture),

d. Enthalpy and temperature of the secondary side fluid,

e. Mass quality and void fraction of the secondary side fluid,

f. Three velocity components of the steam and water phases,

g. Secondary side pressure, and

h. Various other auxiliary parameters.

D.30 EasyFIV

EasyFIV is a PC-based, user-friendly, flow-induced vibration code used for predicting the response of tube bundles (or single tubes) subjected to crossflow. It provides default values and ranges for the constants needed for the analysis. However, some background knowledge in FIV is necessary for the user to judiciously select the constants for the application.

The FIV calculations in EasyFIV are performed in two steps. First, a commercially available, finite-element software (PAL2) integrated within EasyFIV calculates the natural frequencies and mode shapes of the tubes. Then the natural frequency and mode shape information along with the crossflow velocity distribution and other FIV parameters provided by the user are used to predict the FIV response of the tubes. The results of the FIV analysis can be displayed graphically or in tabular form. Hard copies of the tables and graphs can be obtained.

There are four mechanisms that cause tube bundles (subjected to crossflow) to vibrate. They are as follows:

1. Fluid-elastic instability,
2. Vortex shedding,
3. Turbulence buffeting, and
4. Acoustic resonance

EasyFIV predicts the tube response due to the first three mechanisms. Response due to acoustic resonance is not predicted since acoustic resonance is not of concern in liquid and two-phase crossflow situations.

The most common tube geometries that are used for EasyFIV analysis are straight tubes and U-tubes. However, EasyFIV has the capability of modeling any complex-shaped tube geometry. The only restriction is that the entire tube lie in one plane.

D.31 CIRC

The BWC computer code, CIRC provides a one-dimensional, thermal-hydraulic analysis of natural circulation, inverted U-tube nuclear steam generators. Heat transfer, circulation ratio, and water level analysis capabilities are available. The code is capable of analyzing steam generators with integral or nonintegral preheaters. As a minimum, the program determines the thermal performance based on the input parameters. The program can also do partial power cases, specified as a fraction of the full-power steam flow where the primary inlet conditions are recalculated by CIRC.

Inputs vary depending on the scope of analysis required, but basic geometry, terminal point parameters, flow characteristics, and fluid properties are required.

Output for the input case, as well as for specified full-power and partial power cases, includes steam flow rate, heat duty, secondary side fluid qualities and densities, circulation ratio, circulation loop pressure losses, and a water level/inventory analysis.

The BWC thermal-hydraulic analysis program, CIRC, provides a one-dimensional analysis of a natural recirculating inverted U-tube steam generator with either light or heavy water primary fluid. The code has the capability of performing heat transfer, circulation, and water level/inventory analyses.

Heat Transfer Analysis

The CIRC code calculates the heat transferred in the boiler. The basic case of no primary fluid inlet quality and no integral preheater will be considered first. The surface area of the boiling zone (B-Zone) is divided into ten equal area zones and the heat transferred is determined for each zone with the secondary side modeled as an infinite heat sink of constant temperature equal to secondary side saturation. The heat transfer across the tube wall accounts for convection at the tube inside diameter (ID), tube wall metal conductivity, tube outside diameter (OD) fouling, and pool boiling at the tube OD.

The analysis for the case of primary side quality proceeds similarly with the following exceptions. The tube area (A-Zone) required to condense the primary side quality is determined by calculating the heat transfer necessary to reduce the enthalpy of the incoming primary fluid to saturated conditions. The heat transfer coefficient again includes ID convection, metal conductivity, fouling, and boiling coefficient at the tube OD. This area is deducted from the total boiler area input and the remaining area is divided into ten zones. The remainder of the heat transfer analysis proceeds as above.

For the case of the integral preheater, the boiling area heat transfer analysis prior to the preheater is as discussed above. At the preheater, boiling, subcooled boiling, and convective heat transfer zones (C, D, and E Zones, respectively) are considered. The heat transfer analysis in the preheater pool boiling zone is similar to the analysis outside of the preheater. The subcooled boiling region is determined as the region where the boiling heat transfer coefficient is larger than the convective coefficient. In the convective zone, countercurrent heat transfer is assumed (as reflected in the log mean temperature difference) and the input convective heat transfer coefficient at the tube OD is used. The basic solution technique assumes an enthalpy for the secondary fluid leaving the preheater and calculates the corresponding feedwater inlet temperature.

This calculated temperature is compared with the input value, and the outlet enthalpy is adjusted until the calculated value converges to the input feedwater temperature. Effects of leakage through the thermal plate to the hot leg are included in the heat transfer analysis of the convective zone below the thermal plate (F Zone). In this zone, countercurrent convective heat transfer is assumed with a secondary flow equal to the outlet steam flow.

Circulation Analysis

The circulation analysis portion of the CIRC code determines the circulation ratio of the generator. All irrecoverable component pressure losses, as well as static heat, are determined around the circulation loop, based on local fluid properties. When the flow losses equal the static pumping head, a converged solution is obtained.

The circulation analysis first considers the heat transferred in the thermal-hydraulic zones and redistributes it over geometric zones, i.e., tube support spans and U-bend. The fluid properties are then determined at the planes of the tube supports and in the spans between the supports. Slip effects due to the two-phase secondary mixture are considered when calculating pumping head. Values are determined for all dynamic losses (shock, friction, etc.) around the loop and combined with pumping head and static head until convergence is obtained, i.e., circulation losses equal total pumping head. The final circulation ratio is then determined.

During a requested partial power calculation, a water level analysis is performed during each circulation iteration.

Water Level/Inventory Analysis

This segment of the CIRC program determines the steam generator's secondary side inventory or water level for a series of geometric volumes input by the user. The routine works for two modes of input: water level versus power and inventory versus power. For specified water level, the inventory is calculated based on the previously determined average volume densities and the input volumes. For specified inventory, the water level is determined. In addition, for partial power cases, the change in inventory (from 100%) or the new water level can be specified and the unknown quantity is determined. The zero-power water level or inventory is also determined.

D.32 CRAFT2 - Loads Version

CRAFT2 is a thermal-hydraulic code that tracks transient pressures and flows due to system perturbations. The results are processed into forcing functions. The code is used to produce the forces due to pipe breaks in pressurized systems. CRAFT2 is the Framatome Technologies, Inc., version of the original NRC-approved CRAFT code. It was reprogrammed to make the inputs and outputs compatible with other codes.

D.33 COMPAR2

COMPAR2 calculates building compartment pressures resulting from mass and energy input. The code is typically used in asymmetric cavity pressure calculations due to high-energy pipe breaks. These pressures are processed into compartment forcing functions. COMPAR2 is the Framatome Technologies, Inc., version of the NRC-approved COMPAR-MOD1 code (NUREG-0609), which was the original Los Alamos code version. It was reprogrammed to make the inputs and outputs compatible with other codes.

D.34 REFERENCES

1. Portland Cement Association, "Notes on ACI 318-71 Building Code Requirements with Design Applications," pp. 1-14, July 1972.

2. E. L. Wilson, "A Computer Program for the Dynamic Stress Analysis of Underground Structures," Report to U.S. Army Engineering Waterways Experiment Station, January 1965.

3. Newmark, Nathan and Rosenblueth, Emilio. Fundamentals of Earthquake Engineering, Prentice-Hall, Englewood Cliffs, N. J., p. 15, 1971.

4. J. Biggs, Introduction to Structural Dynamics, McGraw-Hill, New York, p. 266, 1964.

5. S. Ghosh and E. Wilson, "Dynamic Stress Analysis of Axisymmetric Structures Under Arbitrary Loading," Report No. EERC 69-10, University of California, Berkeley, September 1969.

6. S. Timoshenko and S. Woinowsky-Krieger, Theory of Plates and Shells, McGraw-Hill, New York, pp. 553-554, 1959.

7. S. Timoshenko and J. Goodier, Theory of Elasticity, McGraw-Hill, New York, 1970.

8. B. Budiansky and P. Radkowski, "Numerical Analysis of Unsymmetric Bending of Shell of Revolution," Journal of the American Institute of Aeronautics and Astronautics, August 1963.

9. H. Reismann and J. Padlog, "Forced, Axisymmetric Motions of Cylindrical Shells," Journal of the Franklin Institute, Vol. 284, No. 5, pp. 308-319, November 1967.

10. S. Klein, "A Study of the Matrix Displacement Method as Applied to Shells of Revolution," Proceedings, Conference on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio, 1965.

11. J. F. Abel, P. P. Cole, D. P. Billington, "Maximum Seismic Response of Cooling Towers," Report No. 73-SM-1, Department of Civil and Geological Engineering Research, Princeton University, March 1, 1973.

12. J. M. Doyle and S. L. Chu, "Liner Plate Buckling and Behavior of Stud and Rip Type Anchors," Proceedings, First International Conference on Structural Mechanics in Reactor Technology, Berlin, Germany, Volume 4, Part H, September 1972.

13. A. G. Young and L. A. Tate, "Design of Liners for Reactor Vessels," Proceedings, Conference on Prestressed Concrete Pressure Vessels, Institute of Civil Engineers, London, 1967, Paper J57.

14. "Nelson Stud Welding Applications in Power Generating Plants," Nelson Stud Welding Company, Lorain, Ohio.

15. S. H. Bush, "Probability of Damage to Nuclear Components Due to Turbine Missiles," Nuclear Safety, 14(3): 187-201 (May-June 1973).

16. D. H. Shaffer, S. C. Chay, D. K. McLain, and B. A. Powell, "Analysis of the Probability of the Generation and Strike of Missiles from a Nuclear Turbine," Westinghouse Electric Corporation, March 1974.

17. Sargent & Lundy, "Turbine Missile Study," Appendix C.O, Byron/Braidwood FSAR, Amendment 17, November 1978.

18. S. N Semanderes, "Methods of Determining the Probability of a Turbine Missile Hitting a Particular Plant Region," Westinghouse Electric Corporation, February 1972.

19. C. K. Wang and C. G. Salmon, Reinforced Concrete Design, Second Edition, Intext Educational Publishers, New York and London, 1973, pp. 436-438.

20. Portland Cement Association, Notes on ACI 318-71 Building Code Requirements with Design Applications, 1972, pp. 9-25 through 9-27.

21. ASME Boiler and Pressure Vessel Code, Section III, 1977, Winter 1979.

22. J. S. Kinney, Indeterminate Structural Analysis, Addison-Wesley Publishing Company, Reading, Massachusetts, 1957, p. 377.

23. "Sample Analysis of a Piping System Class 1 Nuclear," prepared by Working Group on Piping of the Design Subgroup of the Nuclear Power Committee of the ASME Boiler and Pressure Vessel Committee, the American Society of Mechanical Engineers, New York, 1972.

24. ICES DYNAL User's Manual, McDonnell Douglas Automation Co., September 1971.

25. NASTRAN Theoretical Manual, NASA SP-221, September 1970.

26. NASTRAN User's Manual, NASA SP-222, September 1970.

27. Y. K. Cheung and J. D. Davies, "Analysis of Rectangular Tanks," CONCRETE, London, England, Vol. 1, pp. 169-174, May 1967.

28. O. Zienkiwicz and Y. Cheung, "The Finite Element Method for Analysis of Elastic Isotropic and Orthotropic Slabs," Proceedings of the Institute of Civil Engineering, London, England, pp. 471-487, August 1964.

29. P. B. Schnabel et al., "SHAKE: A Computer Program for Earthquake Response Analysis of Horizontally Layered Sites," Report No. EERC 72-12, Earthquake Engineering Research Center, University of California, Berkeley, December 1972.

30. I. M. Idriss and H. B. Seed, "Seismic Response by Variable Damping Finite Elements," Journal of the Geotechnical Engineering Division, ASCE, Vol. 100, No. GT1, Proc. Paper 10284, pp. 1-13, January 1974.

31. H. B. Seed and I. M. Idriss, "Soil Moduli and Damping Factors for Dynamic Response Analyses," Report No. EERC 70-10, Earthquake Engineering Research Center, University of California, Berkeley, December 1970.

32. "ICES SLOPE - Slope Stability Analysis System," McDonnell Douglas Automation Company, 1974.

33. E. L. Wilson, "SAP-A General Structural Analysis Program," SESM Report 70-20, Dept. of Civil Engineering, University of California, Berkeley, 1970.

34. S. Timoshenko and J. Gere, Mechanics of Materials, Van Nostrand Reinhold Company, New York, pp. 99-100, 1972.

35. S. Timoshenko and S. Woinowski-Krieger, Theory of Plates and Shells, McGraw-Hill, New York, p. 118, 1959.

36. K. J. Bathe, E. L. Wilson and F. E. Peterson, "SAPIV, A Structural Analysis Program for Static and Dynamic Response of Linear Systems," Earthquake Engineering Research Center, Report No. EERC 73-11, June 1973.

37. "ADL Pipe Static-Thermal-Dynamic Pipe Stress Analysis," Arthur D. Little, Inc., Cambridge, Massachusetts, January 1971.

38. "Construction Industry Programs, PIPDYN: Dynamic Analysis of Piping Systems," Computer Sciences Corporation, Los Angeles, California.

39. H. Reismann and J. Padlog, "Forced, Axisymmetric Motions of Cylindrical Shells," Journal of the Franklin Institute, Vol. 284, No. 5, Nov., 1967.

40. "A Program to Perform Stress Analysis of Shells of Revolution," Knolls Atomic Power Laboratory, Schenectady, New York, September 1963.

41. J. H. Percy, D. R. Navaratna, S. Klein, "Stress and Bending Analysis for Shells of Revolution III," ASRL TR 121-7, MIT Aeroelastic and Structures Laboratory.

42. Manual of Steel Construction, Seventh Edition, American Institute of Steel Construction, New York, N. Y.

43. R. D. Logcher et al., "ICES STRUDL II, The Structural Design Language Engineering User's Manual," Department of Civil Engineering, M.I.T., 1st ed., November 1968.

44. "ICES STRUDL Improvements," McDonnell Douglas Automation Company, February 1973.

45. "ICES Application Brief," UNIVAC Marketing Support, Sperry Rand Corporation, 1972.

46. R. L. Taylor and C. B. Brown, "Darcy's Flow Solutions with a Free Surface," Journal of the Hydraulics Division, ASCE, Vol. 93, No. HY2, March 1967, pp. 25-33.

47. M. E. Harr, Groundwater and Seepage, McGraw-Hill Book Company, New York 1962.

48. V. T. Chow, Handbook of Applied Hydrology, McGraw-Hill Book Company, New York, 1964.

49. ACI 318-77 Building Code Requirements for Reinforced Concrete Section 9.3, page 29.

50. ASME Boiler and Pressure Vessel Code, Section III Nuclear Power Plant Components, Division 2, Tables CC-3421-1, CC-3431-1, 80, page 170.

51. Bush, S. H., "Probability of Damage to Nuclear Components Due to Turbine Missiles," Nuclear Safety, 14(3): 187-201 (May-June 1973).

52. Shaffer, D. H., Chay, S. C., McLain, D. K. and Powell, B. A., "Analysis of the Probability of the Generation and Strike of Missiles from a Nuclear Turbine," Westinghouse Electric Corporation, March, 1974.

53. Sargent & Lundy, "Turbine Missile Study," Appendix C.0, Byron/Braidwood PSAR, Amendment 12, May, 1975.

54. Semanderes, S. N., "Methods of Determining the Probability of a Turbine Missile Hitting a Particular Plant Region," Westinghouse Electric Corporation, February, 1972.

55. Sheron, Brian W. (NRC) to Cloud, Robert L. (Robert L. Cloud & Associates, Inc.), letter, subject: Topical Report Review of RLCA Report, "A Topical Report on the Methodology, Verification and Applications of Computer Program GAPPIPE," RLCA/P94/04-94/009, June 1, 1994," dated April 11, 1995.

TABLE D-1

SPAN 1 CHARACTERISTICS AND DESIGN RESULTS

		LEFT SIDE	MIDDLE	RIGHT SIDE
Clear span (ft)			23.0	
Section (in.)			24.0 x 36.0	
Design moment M_u (kip-ft)		1130.70	650.00	1204.70
Design shear V_u (kip)		345.40	134.10	230.70
Required area (in^2)	CBEAM	8.62	4.57	9.31
	Hand Calcs.	8.58	4.72	9.36
Required bars	CBEAM	2 - #10 4 - #11	3 - #11	2 - #10 5 - #11
	Hand Calcs.	2 - #10 4 - #10	3 - #11	2 - #10 5 - #11
Provided steel	CBEAM	8.78	4.68	10.34
	Hand Calcs.	8.78	4.68	10.34
Stirrups	CBEAM	#5 - @ 7.0 in.*	#4 - @ 14.0 in.**	#4 - @ 4.0 in.**
	Hand Calcs.	#5 - @ 7.0 in.*	#4 - @ 14.0 in.**	#4 - @ 4.0 in.**

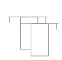
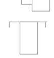
*Type 2 Stirrups: 
**Type 1 Stirrups: 

TABLE D-2

SPAN 2 CHARACTERISTICS AND DESIGN RESULTS

		LEFT SIDE	MIDDLE	RIGHT SIDE
Clear span (ft)			15.5	
Section (in.)			24.0 x 27.0	
Design moment M_u (kip-ft)		627.40	484.30	543.90
Design shear V_u (kip)		132.90	70.40	103.60
Required area (in^2)	CBEAM	6.51	4.77	5.42
	Hand Calcs.	6.69	4.73	5.45
Required bars	CBEAM	2 - #10 5 - #11	4 - #11	6 - #10
	Hand Calcs.	2 - #10 5 - #11	4 - #11	6 - #10
Provided steel	CBEAM	10.34	6.24	7.62
	Hand Calcs.	10.34	6.24	7.62
Type 1 stirrups	CBEAM	#4 - @ 6.0 in.	#4 - @ 12.0 in.	#4 - @ 12.0 in.
	Hand Calcs.	#4 - @ 6.0 in.	#4 - @ 12.0 in.	#4 - @ 12.0 in.

TABLE D-3

SPAN 3 CHARACTERISTICS AND DESIGN RESULTS

		LEFT SIDE	MIDDLE	RIGHT SIDE
Clear span (ft)			15.5	
Section (in.)			2.24 x 27.0	
Design moment M_u (kip-ft)		586.30	503.10	490.40
Design shear V_u (kip)		111.80	67.60	112.80
Required area (in^2)	CBEAM	5.88	4.97	4.84
	Hand Calc.	5.86	4.98	4.86
Required bars	CBEAM	6 - #10	4 - #11	4 - #10
	Hand Calcs.	6 - #10	4 - #11	4 - #10
Provided steel	CBEAM	7.62	6.24	5.08
	Hand Calcs.	7.62	6.24	5.08
Type 1 stirrups	CBEAM	#4 - @ 10.0 in.	#4 - @ 12.0 in.	#4 - @ 9.0 in.
	Hand Calcs.	#4 - @ 10.0 in.	#4 - @ 12.0 in.	#4 - @ 9.0 in.

TABLE D-4

RESULTS FOR 28-DAY STRENGTH

Number of Samples Collected - - -	46				
Mean Observed Strength - - -	3456.1	Allow. Design Str. -	2955.1	Spec. Design Str. -	2500.0
Observed Standard Deviation - - -	373.0	C.O.V. % - - -	10.8	Expected C.O.V. % -	15.0
Within Test Std. Deviation - - -	.0	C.O.V. % - - -	.0	Expected C.O.V. % -	5.0
Mean Observed Range - - - - -	.0				

Total Number of Bad Samples - Average Strength 0

Number of Times Inefficient Testing Observed 0

CONTROL ACCORDING TO ACI MANUAL

Number of Samples Falling Below FC - - - - - - - - - - -	0	Percent of Samples Collected -	.00
Number of Samples Falling Below FC-500 - - - - - - - - -	0	Percent of Samples Collected -	.00
Number of Times Moving Avg. Fell Below FC - - - - - - -	0	Percent of Samples Collected -	.00

TABLE D-5

STRUCTURAL FREQUENCIES

MODE NUMBER	STRUCTURAL FREQUENCY (cps) BIGGS	DYNAS
1	1.00	1.00
2	2.18	2.18
3	3.18	3.18

TABLE D-6

PROBABLE MAXIMUM STORY DISPLACEMENTS

MODE NUMBER	PROBABLE MAXIMUM STORY DISPLACEMENT (inches)	
	BIGGS	DYNAS
1	1.50	1.51
2	3.22	3.20
3	4.86	4.68

TABLE D-7

ABSOLUTE MAXIMUM STORY SHEARS

MODE NUMBER	ABSOLUTE MAXIMUM STORY SHEAR (kips)	
	BIGGS	DYNAS
1	3020	3010
2	2080	2068
3	1345	1353

TABLE D-8

PROBABLE MAXIMUM STORY SHEARS

	PROBABLE MAXIMUM STORY SHEARS (kips)	
MODE NUMBER	BIGGS	DYNAS
1	2550	2262
2	1740	1757
3	895	902

TABLE D-9

NATURAL PERIODS FOR THE EIGHT LOWEST

FLEXURAL MODES

MODE NUMBER	PERIODS (seconds) SAPIV	DYNAS
1	525.7900	525.69
2	85.36800	85.369
3	30.9650	30.964
4	16.0590	16.060
5	9.9006	9.9010
6	6.8276	6.8279
7	5.1865	5.1866
8	4.3777	4.3778

TABLE D-10

COMPUTER OUTPUT

DISPLACEMENTS

LOCATION	VALUE (inches)
U_1	.059492
U_2	.045083
U_3	.033292
U_4	.023913
U_5	.016642
U_6	.011246
U_7	.007491
U_8	.004830
U_9	.002874
U_{10}	.001338
f_1	16.293 kips

Post-Buckling Load

P = 21.978 kips

TABLE D-11

COMPUTER OUTPUT

ANCHOR FORCES

LOCATION	VALUE (kips)
f_1	16.270
f_2	15.430
f_3	14.149
f_4	12.338
f_5	10.935
f_6	9.531
f_7	6.348
f_8	4.093
f_9	2.436
f_{10}	1.134

TABLE D-12

COMPARISON OF P_2 VALUES FROM MISLODS AND SEMANDERES

MISSILE INITIAL VELOCITY RANGE (ft/sec)	MISSILE LOCATION	TARGET A1		TARGET A2		TARGET A3	
		MISLODS	SEMANDERES	MISLODS	SEMANDERES	MISLODS	SEMANDERES
270-330	Interior	3.5E-3*	1.8E-3	1.3E-4	8.7E-5	1.3E-4	7.7E-5
	End	4.6E-4	4.8E-3	5.3E-5	4.1E-5	5.3E-5	3.5E-5
540-660	Interior	8.4E-4	2.7E-4	8.2E-6**	4.5E-6	8.2E-6	4.1E-6
	End	1.1E-4	5.9E-5	3.2E-6	2.0E-6	3.3E-6	3.9E-5

* The notation "3.5E-3" means 3.5×10^{-3} or 0.0035.

** Hand calculation of this value is 8.17E-6.

TABLE D-13

COMPARISON OF MOMENTS FOR SELECTED MEMBERS

	MOMENTS FROM REFERENCE 22 (kip-ft)	MOMENTS FROM PIPSYS (kip-ft)
M_{AB}	106.0	102.8
M_{BA}	72.0	72.5
M_{BC}	133.0	131.8
M_{CB}	133.0	131.8
M_{CD}	-133.0	-131.8
M_{DC}	-133.0	-131.8
M_{DE}	133.0	131.8
M_{ED}	86.0	84.2
M_{BE}	-158.0	-156.6
M_{EB}	-158.0	-156.6
M_{FE}	106.0	102.8
M_{EF}	72.0	72.5

TABLE D-14

SUMMARY OF LOAD SETS AT GIRTH BUTT-WELD WITH CHANGE IN MATERIAL AND

WALL THICKNESS, LOCATION 19

LOAD SET NO.	LOAD SET DESCRIPTION	NO. OF TRANSIENTS	P	M_x	M_y	M_z	ΔT_1	T_a (VALVE)	T_b (PIPE)	ΔT_2
1	Zero	) 5	0	0.0	0.0	0.0	0.0	70	70	0.0
2	Cold Hydro Test		3590	0.0	0.0	0.0	0.0	70	70	0.0
3	Hot Hydro Test, Up	) 40	2200	251.7	141.6	-7.1	2.4	400	400	0.3
4	Hot Hydro Test, Down		0	0.0	0.0	0.0	-2.4	70	94	-0.3
5	Plant Startup	) 100	2200	337.2	184.9	-936.0	0.0	70	70	0.0
6	Plant Shutdown		0	0.0	0.0	0.0	0.0	70	70	0.0
7	Plant Loading	) 18300	2200	381.6	204.4	-1169.6	0.0	70	70	0.0
8	Plant Unloading		2200	337.2	184.9	-936.0	0.0	70	70	0.0
9	Loss of Load, 4.1	) 80	2515	384.2	204.4	-1183.4	0.0	70	70	0.0
10	Loss of Load, 4.2		1500	345.7	186.4	-1011.4	0.0	70	70	0.0
11	N.O. + Earthquake	) 50	2200	408.6	463.3	-1134.1	0.0	70	70	0.0
12	N.O. - Earthquake		2200	265.8	-93.5	-737.9	0.0	70	70	0.0

TABLE D-15

SIX HIGHEST VALUES OF STRESS INTENSITY, GIRTH BUTT WELD WITH

CHANGE IN MATERIAL AND WALL THICKNESS

		VALUES FROM REFERENCE 23				PIPSYS PROGRAM			
LOAD SET PAIR		S_n	Eq. (12)	Eq. (13)	K_e	S_n	Eq. (12)	Eq. (13)	K_e
3	4	52549	(*)	(*)	1.000	52600	(*)	(*)	1.000
3	9	49883	(*)	(*)	1.000	49900	(*)	(*)	1.000
3	10	49620	(*)	(*)	1.000	49600	(*)	(*)	1.000
3	6	48013	(*)	(*)	1.000	48000	(*)	(*)	1.000
1	3	48013	(*)	(*)	1.000	48000	(*)	(*)	1.000
3	11	47728	(*)	(*)	1.000	47700	(*)	(*)	1.000

(*)Because S_n, calculated by Equation (10), is less than $3S_m$, Equations (12) and (13) are satisfied.

TABLE D-16

SUMMARY OF CALCULATIONS OF CUMULATIVE USAGE FACTOR, GIRTH BUTT WELD

WITH CHANGE IN MATERIAL AND WALL THICKNESS

LOAD SET PAIR		VALUES BASED ON REFERENCE 23		VALUES FROM PIPSYS PROGRAM	
i	j	$\frac{S_p K_e}{2}$	USAGE FACTOR	$\frac{S_p K_e}{2}$	USAGE FACTOR
3	9	40338	0.0050	40300	0.005
4	9	34400	0.0029	34400	0.003
1	11	29806	0.0002	29800	0.000
6	11	29806	0.0020	29800	0.002
6	7	29163	0.0023	29200	0.002
2	10	26254	0.0002	26300	0.000
10	12	93170	0.0000	93200	0.000
Cumulative Usage Factor			0.0126		0.0124

TABLE D-17

MODAL FREQUENCIES
(cycle/sec)

MODE NO.	PIPSYS	NASTRAN	DYNAL
1	6.07	6.085764	6.0821088
2	10.69	10.94144	10.936468
3	11.48	11.66862	11.666215
4	14.76	15.20947	15.204282
5	20.12	22.25613	22.135260
6	23.87	28.53255	28.505264
7	25.32	30.58105	30.530972
8	28.80	31.22073	31.190062
9	30.00	32.27319	32.199679
10	42.39	43.14653	43.135100
11	42.95	43.50436	43.497053
12	58.02	58.19336	57.991710
13	77.78	76.62025	71.996751
14	90.74	93.69710	92.12974
15	91.8	96.04482	95.167976
16	93.39	97.81956	97.410131
17	96.96	99.40727	98.209594
18	101.42	104.6169	101.64513
19	102.14	105.4910	103.80206
20	103.03	107.7136	107.52304

TABLE D-18

ALLOWABLE SHEAR, MOMENT AND SPAN OF CABLE TRAY

SHEAR, SPAN, OR MOMENT	SEISHANG	HAND CALCULATION
Vertical shear, static (kip)	16.05	16.05
Positive bending moment, static (k-in.)	50.64	50.83
Negative bending moment, static (k-in.)	57.62	57.64
Vertical shear, seismic (kip)	20.84	20.81
Horizontal shear, seismic (kip)	12.84	12.83
Positive bending moment, seismic (k-in.)	67.51	67.61
Negative bending moment, seismic (k-in.)	76.83	76.82
Horizontal bending moment, seismic (k-in.)	153.61	153.59
Span (ft)	20.78	20.75

TABLE D-19

RESPONSE OF THE CEILING-MOUNTED SUPPORT

		SEISHANG	DYNAS
Horizontal period (sec)		0.1742	0.1765
Vertical period (sec)		0.0092	0.0093
Forces and moments due to horizontal seismic			
Vertical element (No. 1)	axial (lb)	1600	1607
	shear (lb)	770	772
	bending (lb-in.)	17100	17208
Horizontal element (No. 9)	axial (lb)	25	26
	shear (lb)	302	304
	bending (lb-in.)	10900	10944
Forces and moments due to vertical seismic			
Vertical element (No. 1)	axial (lb)	383	340
	shear (lb)	0	0
	bending (lb-in.)	30	24
Forces and moments due to dead load			
Vertical element (No. 1)	axial (lb)	776	774
	shear (lb)	0	0
	bending (lb-in.)	30	0

TABLE D-20

RESPONSE OF THE WALL-MOUNTED SUPPORT

		SEISHANG	DYNAS
Horizontal period (sec)		0.0067	0.0067
Vertical period (sec)		0.1065	0.1080
Forces and moments due to horizontal seismic			
Vertical element (No. 6)	axial (lb)	0	1
	shear (lb)	2	2
	bending (lb-in.)	35	48
Horizontal element (No. 11)	axial (lb)	101	105
	shear (lb)	2	2
	bending (lb-in.)	23	24
Forces and moments due to vertical seismic			
Vertical element (No. 6)	axial (lb)	39	0
	shear (lb)	131	128
	bending (lb-in.)	2700	2676
Forces and moments due to dead load			
Vertical element (No. 1)	axial (lb)	717	702
	shear (lb)	303	329
	bending (lb-in.)	4910	5208

TABLE D-21

INTERACTION COEFFICIENTS OF THE CEILING-MOUNTED SUPPORT

INTERACTION COEFFICIENT	SEISHANG	PIPSYS
Vertical element (No. 2)	0.617	0.620
(No. 5)	0.520	0.516
Horizontal element (No. 6)	0.683	0.678
Brace element (No. 3)	0.569	0.553

TABLE D-22

APPLIED LOADS FOR SLSAP PIPE NETWORK

LOADING TYPE	DIRECTION X	DIRECTION Y	DIRECTION Z
Concentrated:			
At Node 3		1000.0	
At Node 4		-200.0	
At Node 8	3000.0	1000.0	2000.0
Distributed weight		-6284.0	
Total	3000.0	-4484.0	2000.0

TABLE D-23

FORCE EQUILIBRIUM REACTIONS

	SLSAPIV			SAPIV			ADLPIPE		
NODE	FX	FY	FZ	FX	FY	FZ	FX	FY	FZ
9	5643.5	-	-	5643.51	-	-	5659.0	-	-
11	-	-4044.7	-	-	-4044.59	-	-	-4052.0	-
12	2350.1	4023.1	-4960.9	2350.08	4023.01	-4960.70	2361.0	4026.0	-4966.0
13	-10993.5	4505.6	2960.6	-10993.59	4505.61	2960.70	-11021.0	4509.0	2966.0
TOTAL	-2999.9	4484.0	-2000.3	-3000.00	4484.03	-2000.00	-3001.0	4483.0	-2000.0

TABLE D-24

PERIODS OF PLANE FRAME

MODE NUMBER	PERIOD SLSAPIV (sec)	PERIOD SAPIV (sec)
1	8.182	8.183
2	2.673	2.673
3	1.543	1.543

TABLE D-25

COMPARISON OF MOMENT

(SLSAPIV AND PIPDYN)

ELEMENT NUMBER	MOMENT MZ (kip/in) IN ELEMENT LOCAL COORDINATES (AT ELEMENT END I)		
	SLSAPIV	SAPIV	PIPDYN
1	376.9	376.9	377.0
2	30.66	30.67	30.68
3	152.9	152.9	152.9
4	100.6	100.6	100.6
5	83.27	83.27	83.27
6	46.17	46.17	46.19
7	1.081	1.081	1.082
8	21.59	21.59	21.81
9	7.052	7.052	7.038
10	7.537	7.537	7.571
11	160.3	160.3	160.4
12	78.07	78.07	78.09
13	26.08	26.08	25.80

TABLE D-26

CANTILEVER BEAM ANALYSIS -

NATURAL PERIODS FOR THE EIGHT

LOWEST FLEXURAL MODES

MODE NUMBER	PERIOD SLSAPIV (sec)	PERIOD SAPIV (sec)
1	525.8	525.79
2	85.37	85.368
3	30.96	30.965
4	16.06	16.059
5	9.901	9.9006
6	6.828	6.8276
7	5.186	5.1865
8	4.378	4.3777

TABLE D-27

CYLINDRICAL TUBE ANALYSIS:

SELECTED NATURAL PERIODS

MODE NUMBER	PERIOD SLSAPIV (sec x 10^{-3})	PERIOD SAPIV (sec x 10^{-3})
1	1.279	1.2788
5	0.6214	0.62140
10	0.3298	0.32983
15	0.1746	0.17463
20	0.1150	0.11497

TABLE D-28

ROLLED BEAM DESIGN PROBLEM

	MAXIMUM MOMENTS (kip-ft)	SECTION SELECTED	SECTION MODULUS (in^3)
AISC	125.	W16x40	64.6
STAND	125.58	W18x40	68.4

TABLE D-29

COMPOSITE BEAM DESIGN PROBLEM

	BENDING MOMENTS (kip-ft)		MAXIMUM		NUMBER OF
	CONSTRUCTION LOAD	DESIGN LOAD	SHEAR (kips)	STEEL SECTION	SHEAR CONNECTORS
AISC	71.3	237.2	26.4	W21x44	42
STAND	71.3	236.5	26.3	W21x44	42

TABLE D-30

COLUMN DESIGN PROBLEM

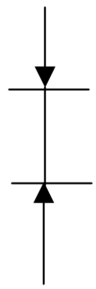
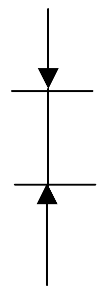
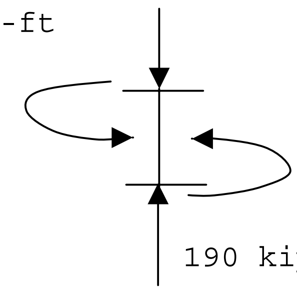
ITEMS	AISC EXAMPLE 1	AISC EXAMPLE 2	AISC EXAMPLE 5
Column Design Parameters	670^{kips}  670^{kips}	540^{kips}  540^{kips}	600^{kips} 100 kip-ft  190 kip-ft 600^{kips}
AISC SOLUTION	W12X161	W12X99	W14x142
STAND SOLUTION	W12x161	W12x99	W14x142

TABLE D-31

PLATE GIRDER DESIGN PROBLEM

RESULTS	AISC	STAND
Maximum bending moment (kip-ft)	2054	2045
Maximum vertical shear (kips)	142	141.3
Web section	1 plate, 70x5/16	1 plate, 70x5/16
Flange section	2 plates, 18x3/4	2 plates, 18x3/4
Stiffener end spacing (ft)	3.4	3.56
Stiffener intermediate spacing (ft)	6.75	6.72
Area of * stiffeners furnished (in^2)	2.0	1.88

*Required area is 1.78 in^2.

TABLE D-32

SECTION AND MATERIAL PROPERTIES

SECTION AND MATERIAL PROPERTIES	PROBLEM NUMBER 1	2	3
Thickness (in.)	42.00	30.00	42.00
Width (in.)	12.00	12.00	12.00
Area of first steel layer (in^2)	6.25	2.25	3.12
Distance of first steel layer (in.)	3.00	3.00	3.00
Area of second steel layer (in^2)	6.25	4.00	3.12
Distance of second steel layer (in.)	37.00	25.00	37.00
Concrete unit weight (lb/ft^3)	150.00	150.00	150.00
Concrete compressive strength (lb/in^2)	4000.00	4000.00	4000.00
Concrete coeff. of thermal expansion (in/°F)	5.56×10^{-6}	5.56×10^{-6}	5.56×10^{-6}
Steel yield strength ($kips/in^2$)	45.00	45.00	45.00
Steel modulus of elasticity ($kips/in^2$)	29000.00	29000.00	29000.00
Material properties	Nonlinear	Nonlinear	Linear
Applied axial force (kips)	-38.25	76.53	34.65
Applied bending moment (ft-kips)	129.75	-9.49	206.25
Inside temperature (°F)	82.50	67.50	247.50
Outside temperature (°F)	52.50	0.00	115.50

TABLE D-33

RESULTS OF TEMCO PROBLEMS

RESULTS	PROBLEM NUMBER 1	2	3
Equilibrating axial force given by TEMCO program (kips)	-38.25	76.53	34.65
Equilibrating axial force computed by hand (kips)	-38.253	76.53	34.65
Equilibrating bending moment given by TEMCO program (ft-kips)	129.75	-9.49	206.26
Equilibrating bending moment computed by hand (ft-kips)	129.752	-9.493	206.25
Thermal moment given by TEMCO program (ft-kips)	-54.58	-21.07	-137.75
Thermal moment computed by hand (ft-kips)	-54.585	-21.071	-137.757

TABLE D-34

INPUT FOR TENSILE FORCE AND

BIAXIAL BENDING PROBLEM

SECTION AND MATERIAL PROPERTIES	PROBLEM 4
Thickness (in.)	42.0
Width (in.)	12.0
Area of each steel bar (in^2)	1.25
Number of steel bars	10.0
Concrete unit weight (lb/ft^3)	150.0
Concrete compressive strength (lb/in^2)	4000.0
Steel yield strength ($kips/in^2$)	45.0
Steel modulus of elasticity ($kips/in^2$)	29000.0
Material properties	Nonlinear
Applied axial force (kips)	21.0
Applied x bending moment (ft-kips)	125.0
Applied y bending moment (ft-kips)	125.0

TABLE D-35

RESULTS FROM TENSILE FORCE AND

BIAXIAL BENDING PROBLEM

RESULTS	PROBLEM 4
Equilibrating axial force given by TEMCO (kips)	20.999
Equilibrating axial force computed by hand (kips)	22.733
Equilibrating x bending moment given by TEMCO (ft-kips)	125.000
Equilibrating x bending moment computed by hand (ft-kips)	124.630
Equilibrating y bending moment given by TEMCO (ft-kips)	125.000
Equilibrating y bending moment computed by hand (ft-kips)	123.753

TABLE D-36

PARAMETERS FOR COLID RECTANGULAR SECTION

STRESS FACTOR EXAMPLE

B	=	12.0 in.
T	=	72.0 in.
X_1	=	10.41 in.
A_{S1}	=	1.56 in^2
X_2	=	69.04 in.
A_{S2}	=	1.56 in^2
F_y	=	60.0 ksi
F_c'	=	4.5 ksi

	STRESS FACTORS		
	STEEL	CONCRETE BENDING	CONCRETE MEMBRANE
Primary and Secondary	0.9	0.850	0.765
Primary	0.4	0.600	0.300

TABLE D-37

COMPARISON OF COLID RESULTS AND HAND CALCULATIONS

FOR RECTANGULAR SECTION STRESS FACTOR EXAMPLE

	HAND CALCULATIONS		COLID	
POINT	Axial Load P_u (kips)	Moment M_u (ft-kips)	Axial Load P_u (kips)	Moment M_u (ft-kips)
1	-168.5	-53.3	-168.5	-52.3
2	3084.0*	- - -	3091.0	0.0
3	967.8	2340.0	968.1	2345.0
4	1058.0	-2395.0	1051.0	-2396.0
5	-74.9	-23.65	-74.9	-23.24
6	1197.0*	- - -	1201.0	0.0
7	404.0	1048.0	404.1	1056.0
8	446.5	-1050.0	413.4	-1053.0

*Maximum Membrane Forces

TABLE D-38

PARAMETERS FOR COLID RECTANGULAR SECTION ULTIMATE CAPACITY AND ACI ULTIMATE CAPACITY OPTIONS

B	=	12.0	in.
T	=	48.0	in.
X_1	=	6.0	in.
A_{S1}	=	1.5	in^2
X_2	=	44.0	in.
A_{S2}	=	2.0	in^2
F_y	=	60.0	ksi
F_c'	=	4.0	ksi

TABLE D-39

COMPARISON OF COLID RESULTS AND HAND CALCULATIONS

FOR RECTANGULAR SECTION ULTIMATE CAPACITY AND

ACI ULTIMATE CAPACITY OPTIONS

	HAND CALCULATIONS		COLID	
	M_u (ft-kips)	P_u (kips)	M_u (ft-kips)	P_u (kips)
Ultimate	1209.0	775.1	1211.0	775.4
ACI Ultimate	846.2	543.3	847.4	542.8

TABLE D-40

PARAMETERS FOR COLID SOLID CIRCULAR COLUMN TEST

Outer	36.0 in.
Diameter of Reinforcement	29.76 in.
Area of Steel	56 in^2
F_y	60.0 ksi
F_c'	5.0 ksi

TABLE D-41

COMPARISON OF COLID RESULTS AND HAND CALCULATIONS

FOR SOLID CIRCULAR COLUMN

	BALANCE POINT		PURE COMPRESSION POINT	
	M_u (ft-kips)	P_u (kips)	M_u (ft-kips)	P_u (kips)
Hand Calculation	2427	1320	0	4170
COLID	2425	1318	0	4170

TABLE D-42

PARAMETERS FOR COLID HOLLOW CIRCULAR COLUMN EXAMPLE

Outer Diameter	36.0 in.
Inner Diameter	24.0 in.
Diameter of Reinforcement	29.76 in.
Area of Steel	64.0 in^2
F_y	6.0 ksi
F_c'	5.0 ksi

TABLE D-43

COMPARISON OF COLID RESULTS AND HAND CALCULATIONS

FOR HOLLOW CIRCULAR COLUMN

	BALANCE POINT		PURE COMPRESSION POINT	
	M_u (ft-kips)	P_u (kips)	M_u (ft-kips)	P_u (kips)
Hand Calculation	2351	898	0	3340
COLID	2351	898	0	3340

TABLE D-44

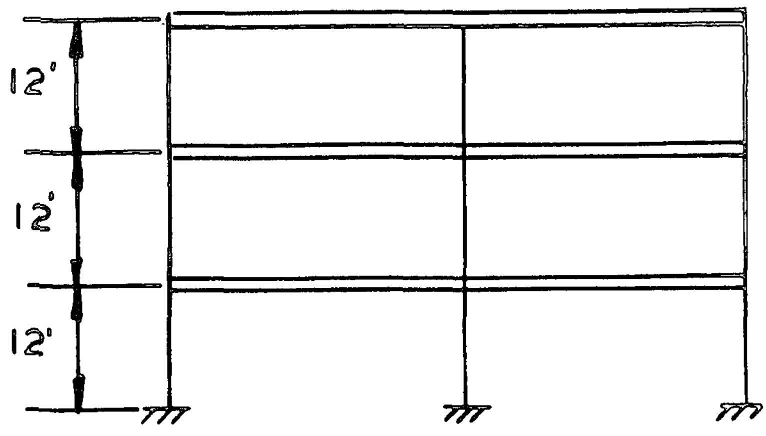
COMPARISON OF COLID RESULTS FOR METRIC AND BRITISH UNITS

ENGLISH		CONVERTED INTO METRIC		METRIC	
M_u (ft-kips)	P_u (kips)	M_u (kg-m)	P_u (kg)	M_u (kg-m)	P_u (kg)
1211.0	775.4	170.0×10^4	351.7×10^3	167.3×10^3	351.5×10^3

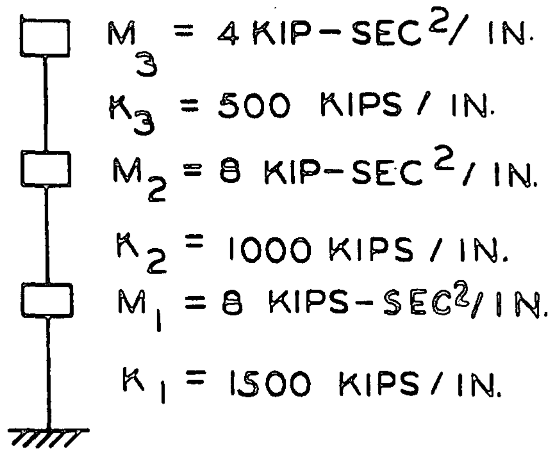
TABLE D-45

ALLOWABLE SLENDERNESS RATIOS

MEMBER TYPE	MAXIMUM SLENDERNESS RATIO (kl/r)
Compression members (verticals, diagonals and longitudinal braces) in floor and wall mounted supports (i.e., compression system supports)	200
Compression members (verticals, diagonals and longitudinal braces) in ceiling mounted supports (i.e., tension system supports)	300



(A)

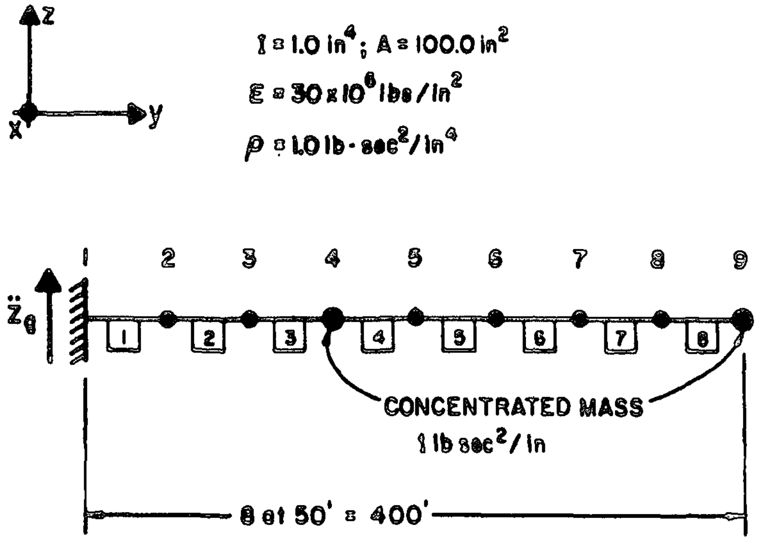


(B)

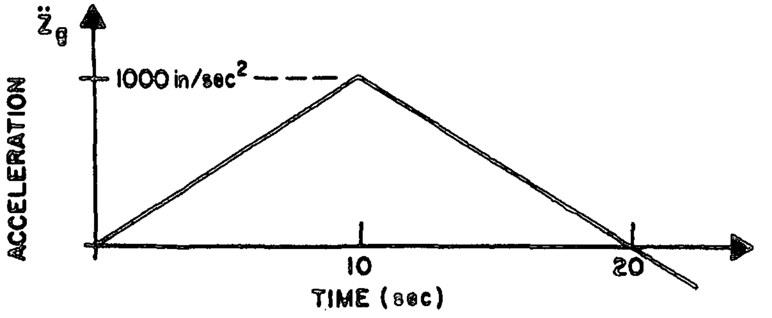
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-1

THREE-STORY SHEAR BUILDING



(a) NODE AND BEAM NUMBER ASSIGNMENTS FOR THE CANTILEVER MODEL

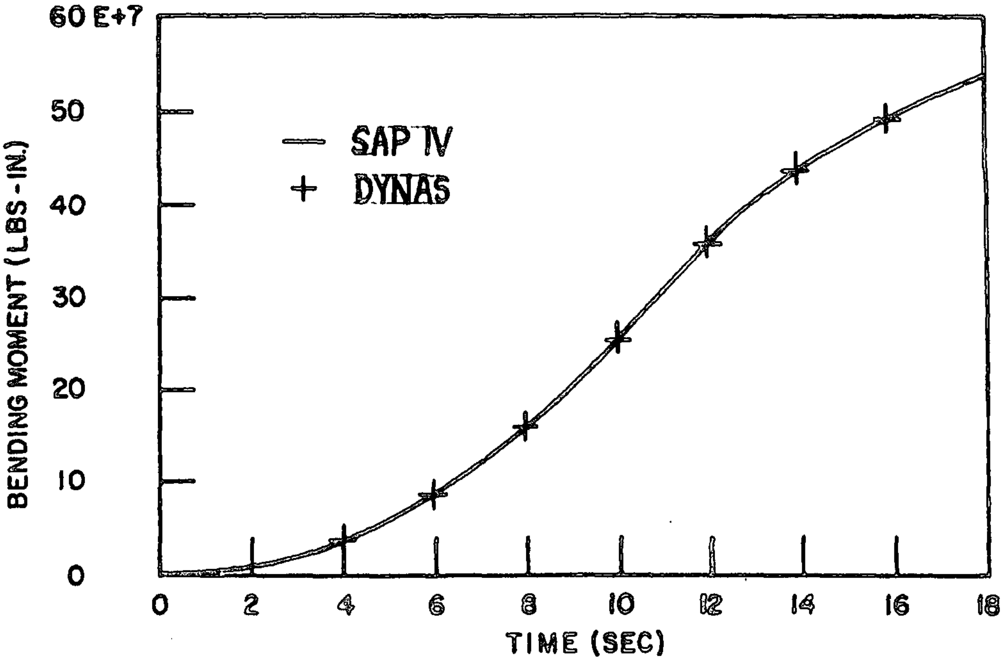


(b) GROUND ACCELERATION APPLIED AT NODE 1

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-2

RESPONSE HISTORY ANALYSIS OF CANTILEVER BEAM FOR DYNAS PROGRAM

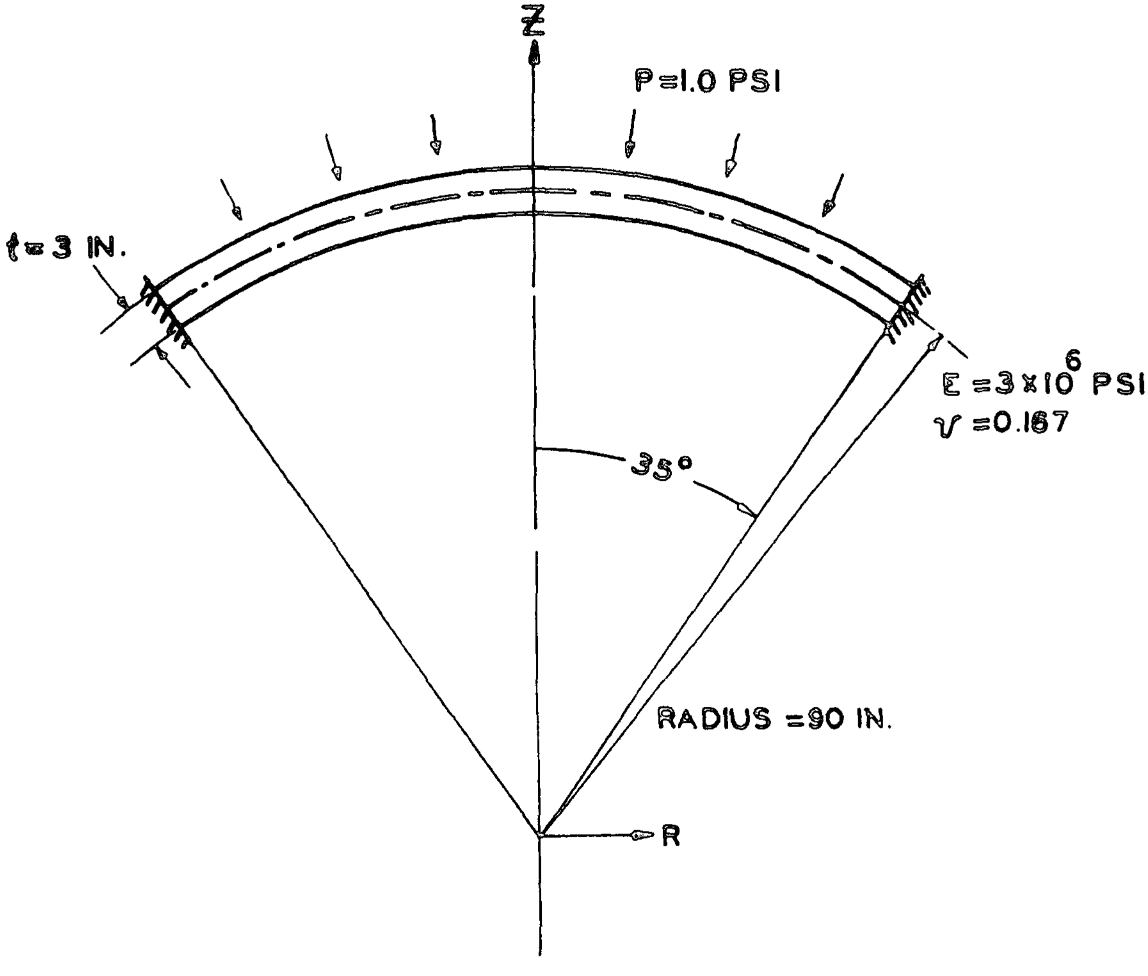


MOMENT AT NODE 1
(FIXED END OF CANTILEVER)

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-3

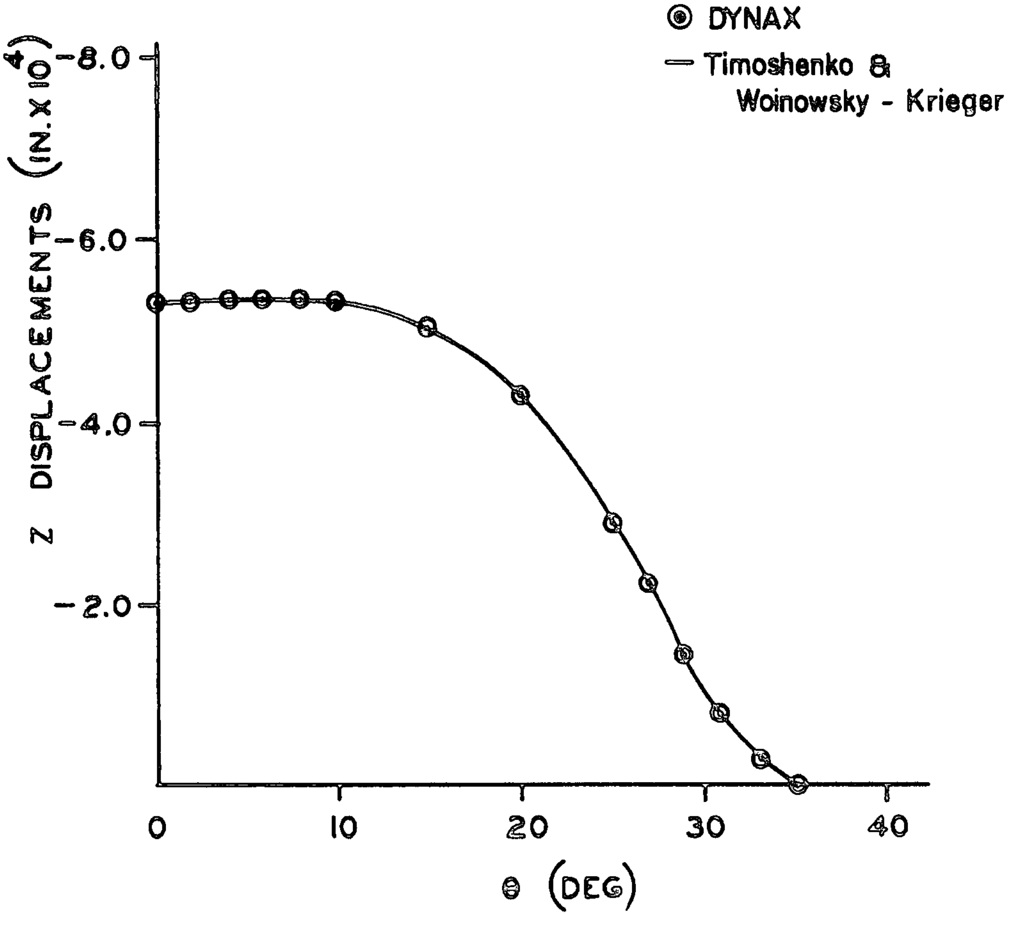
CANTILEVER RESPONSE



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-4

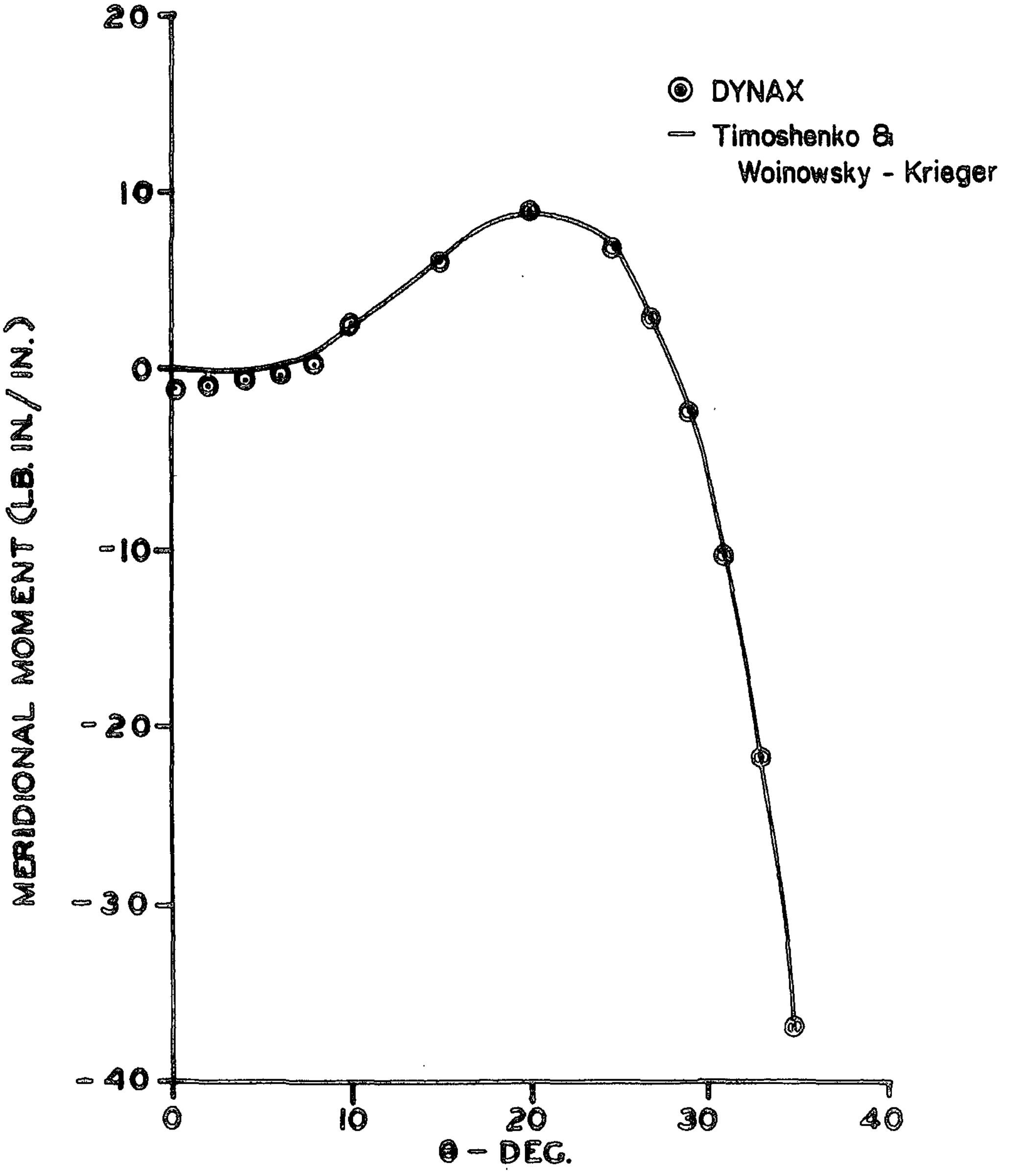
SHALLOW SPHERICAL SHELL



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-5

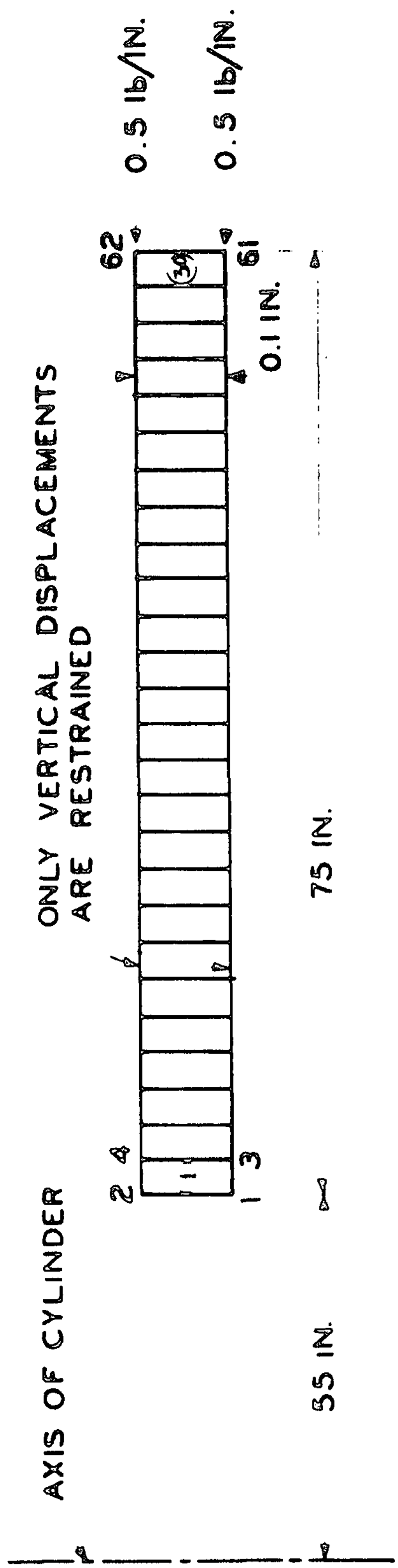
AXIAL DISPLACEMENT SHALLOW SPHERICAL SHELL



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-6

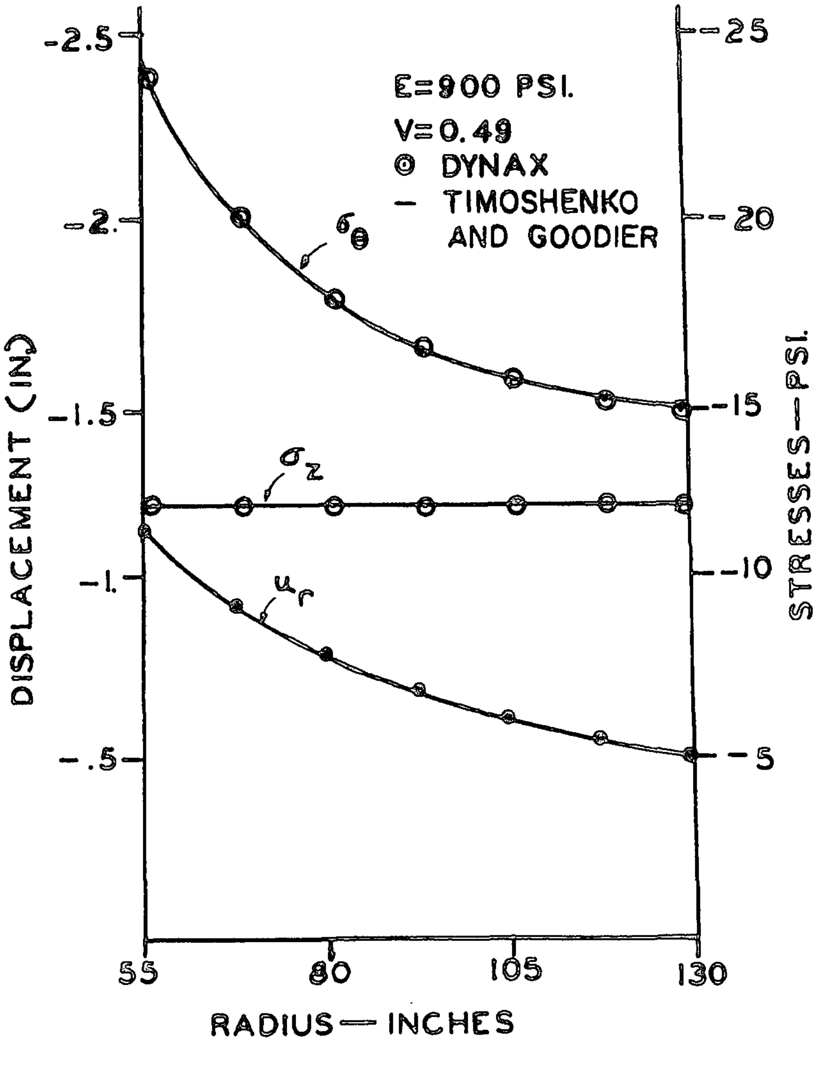
MERIDIONAL MOMENT SHALLOW
SPHERICAL SHELL



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-7

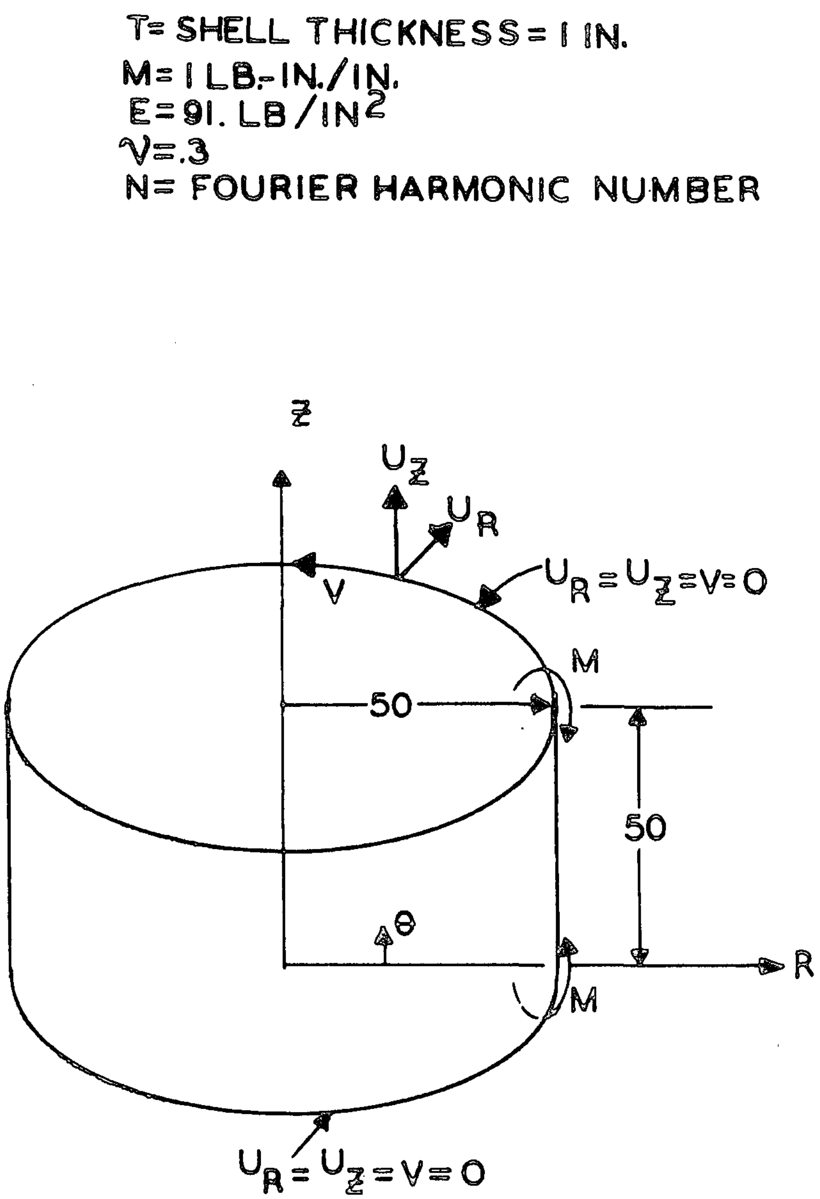
FINITE ELEMENT IDEALIZATION OF
THICK-WALLED CYLINDER



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-8

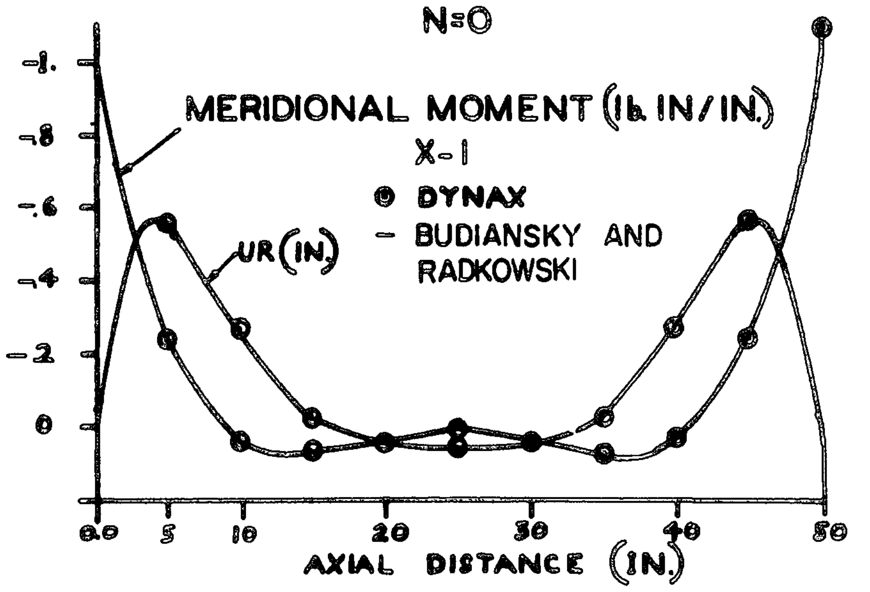
STRESSES AND DISPLACEMENTS IN THICK-WALLED CYLINDERS

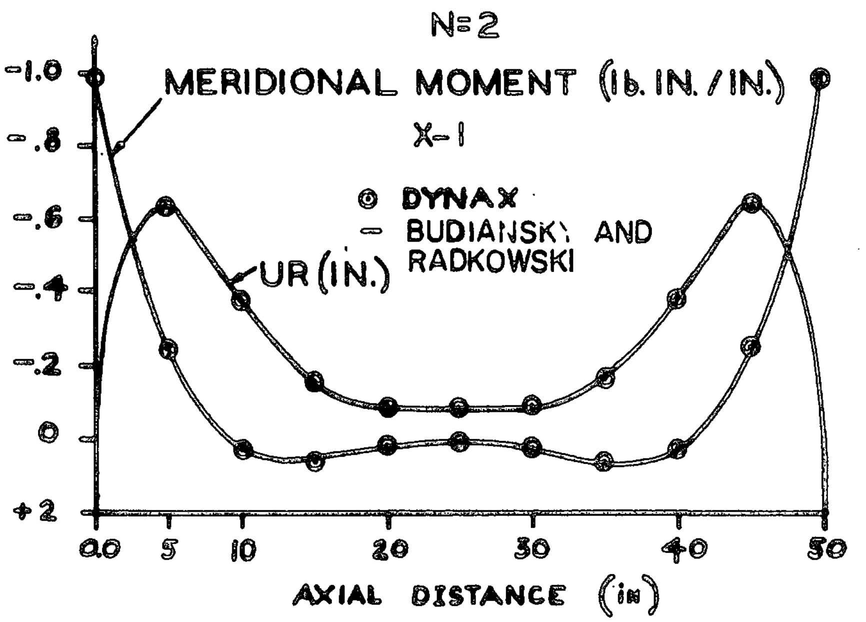


BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-9

CYLINDER UNDER HARMONIC LOADS

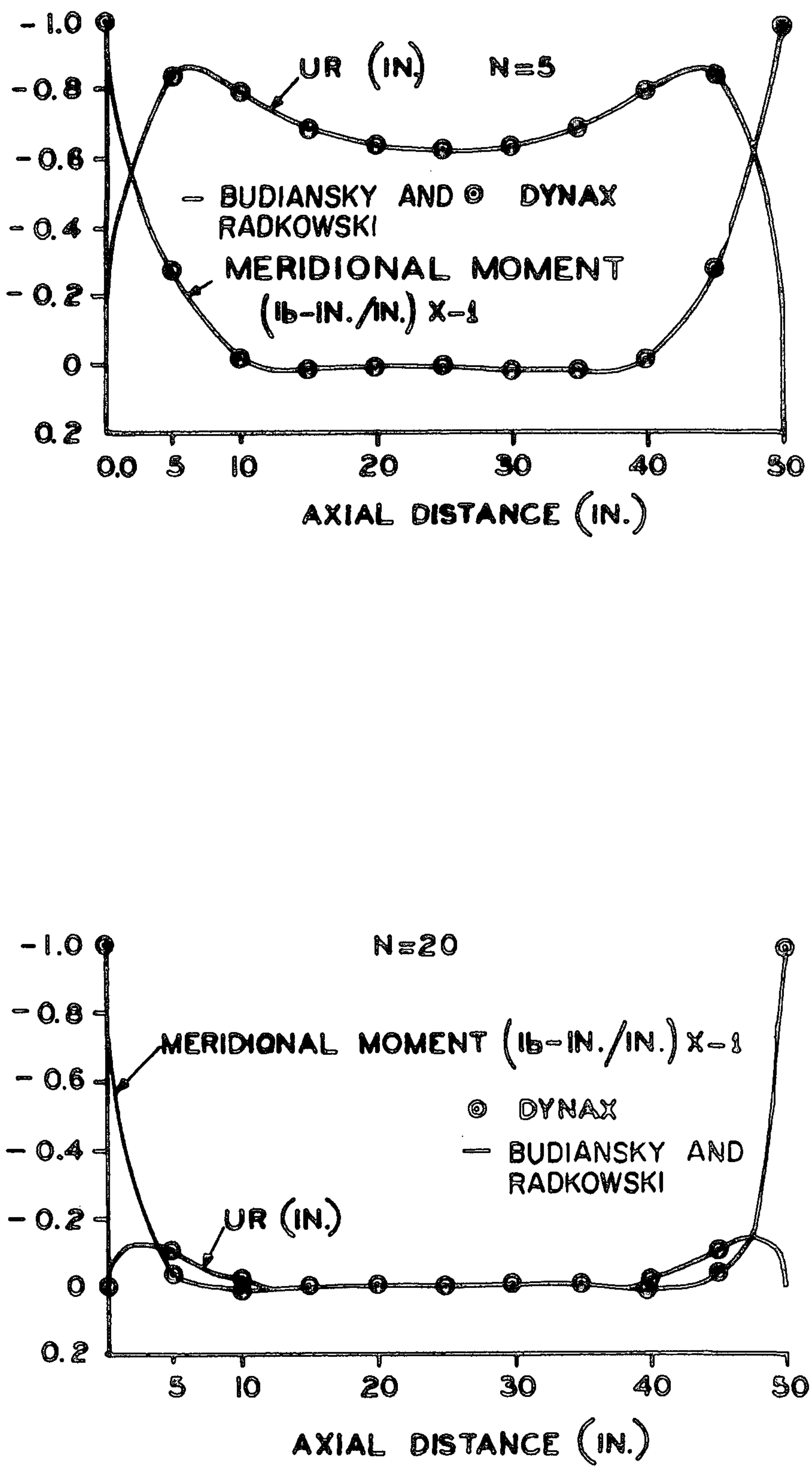




BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-10

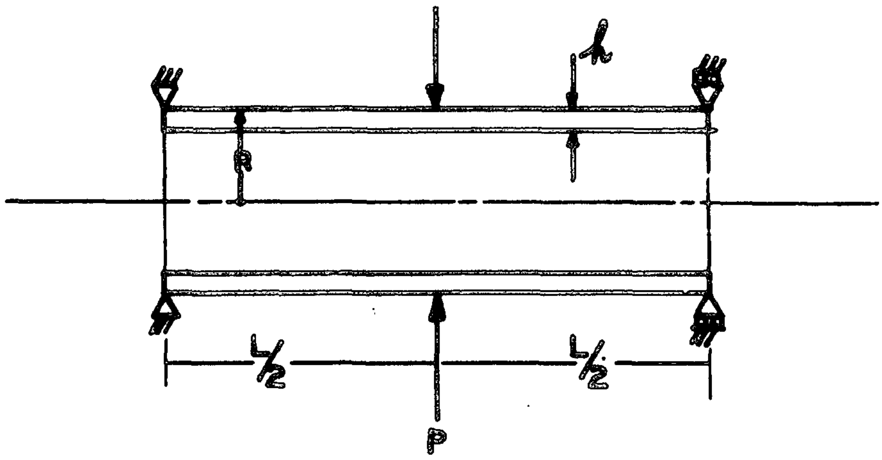
MERIDIONAL MOMENTS AND
DEFLECTIONS OF CYLINDER
N=0 AND N=2

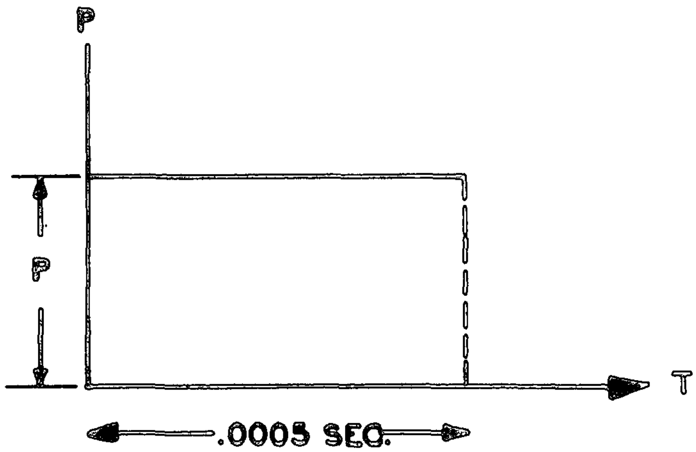


BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-11

MERIDIONAL MOMENTS AND DEFLECTIONS OF CYLINDER N=5 AND N=20





TIME HISTORY LOADING

L = 18 IN. MASS DENSITY (ρ) = 0.0187 $\frac{\#\ SEC.^2}{IN^4}$

P = 500 lb. ν = 0.3

R = 3 IN.

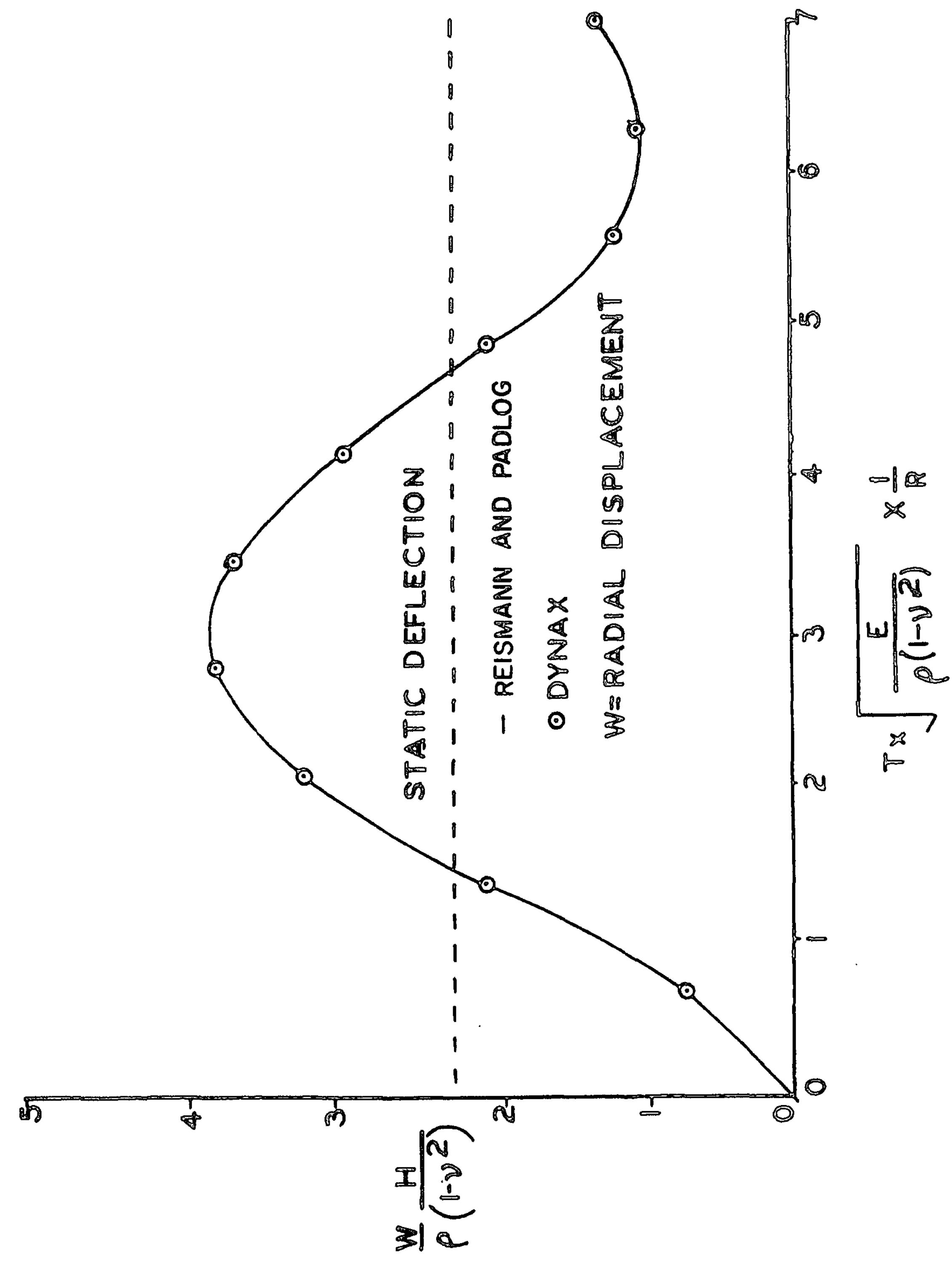
h = 0.3 IN. TIME STEP = .000005 SEC.

E = 30X10^6 lb./IN.^{2}

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-12

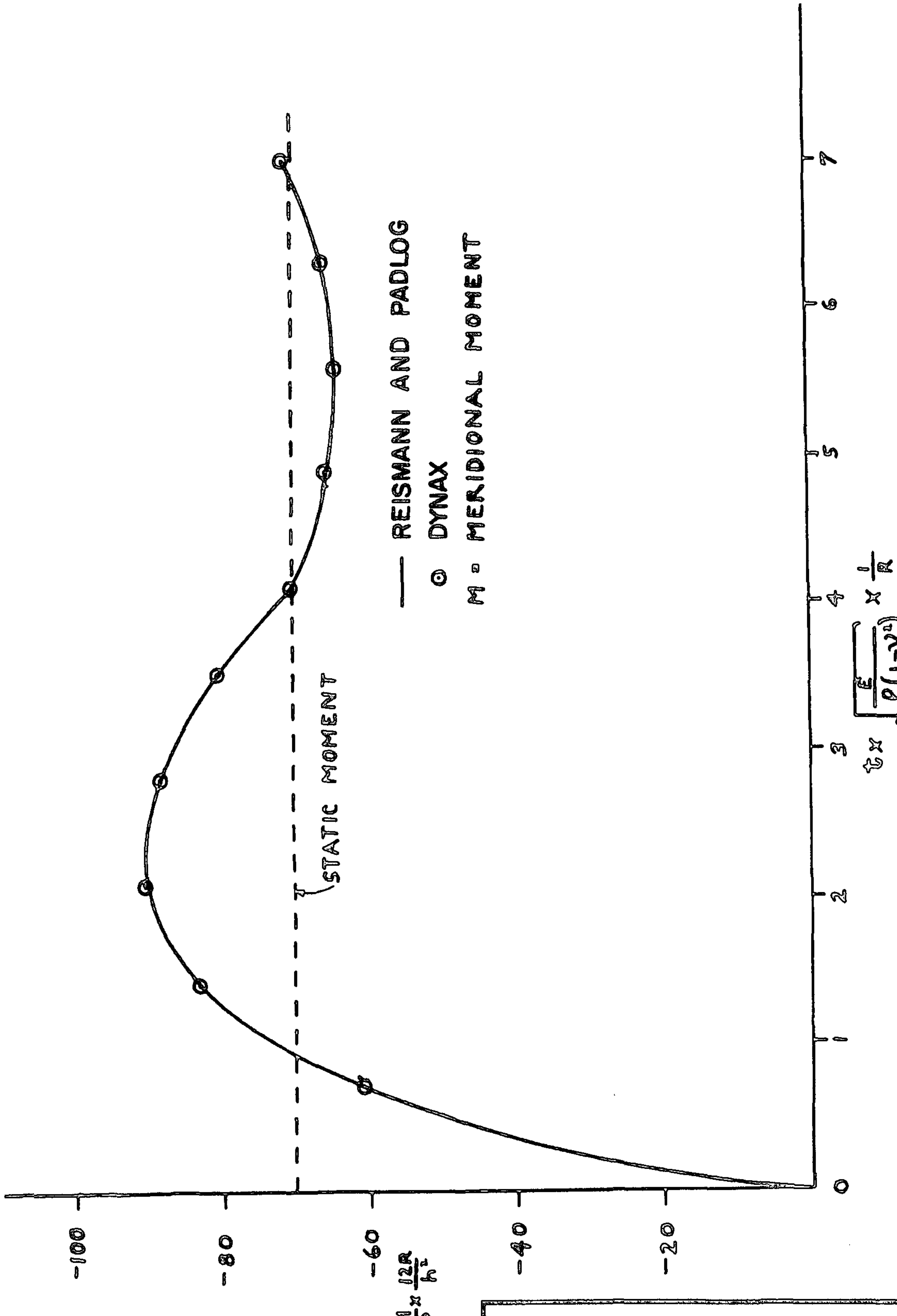
SUDDENLY APPLIED RING LINE LOAD



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-13

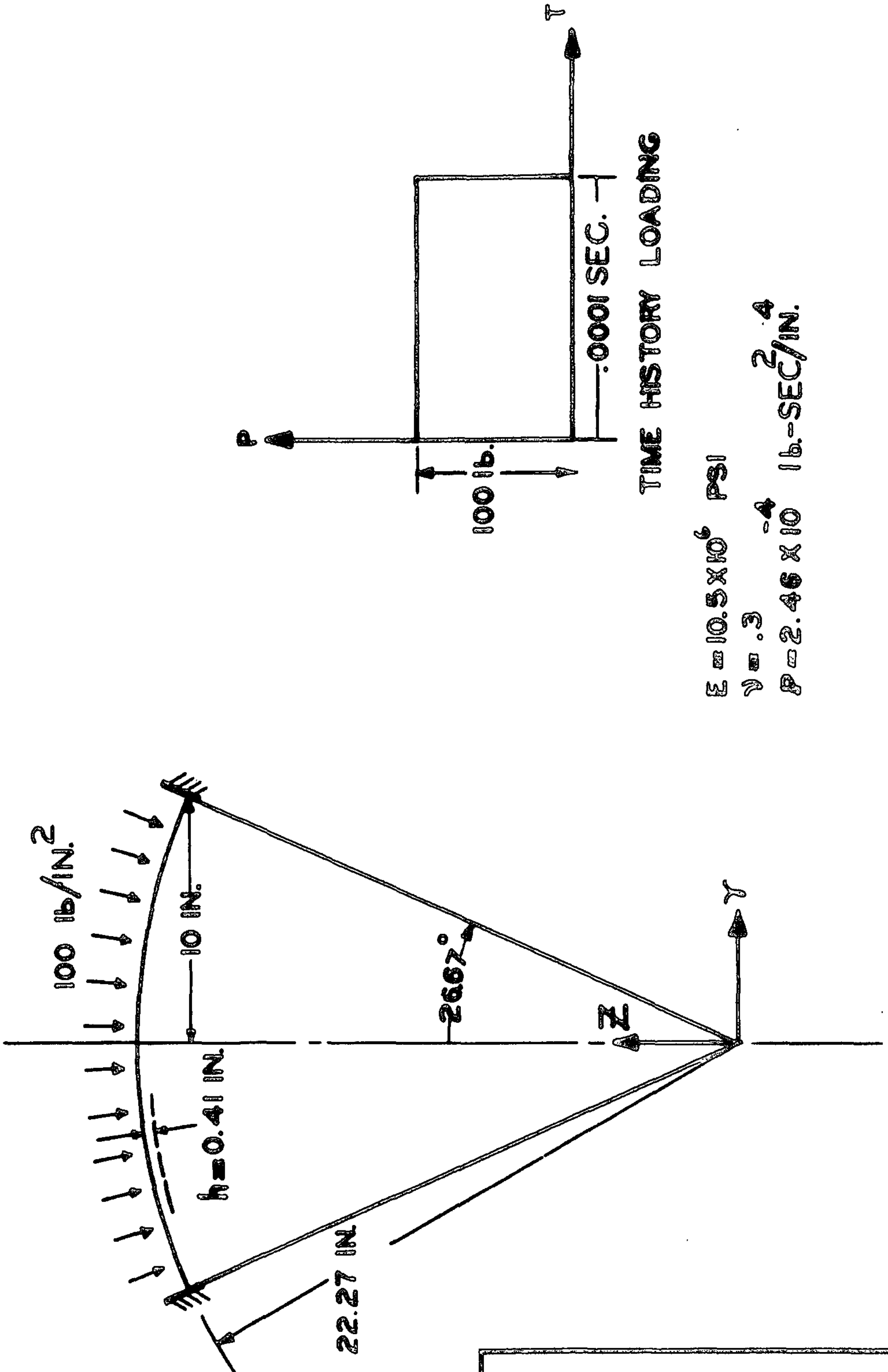
RADIAL DISPLACEMENT VS. TIME



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-14

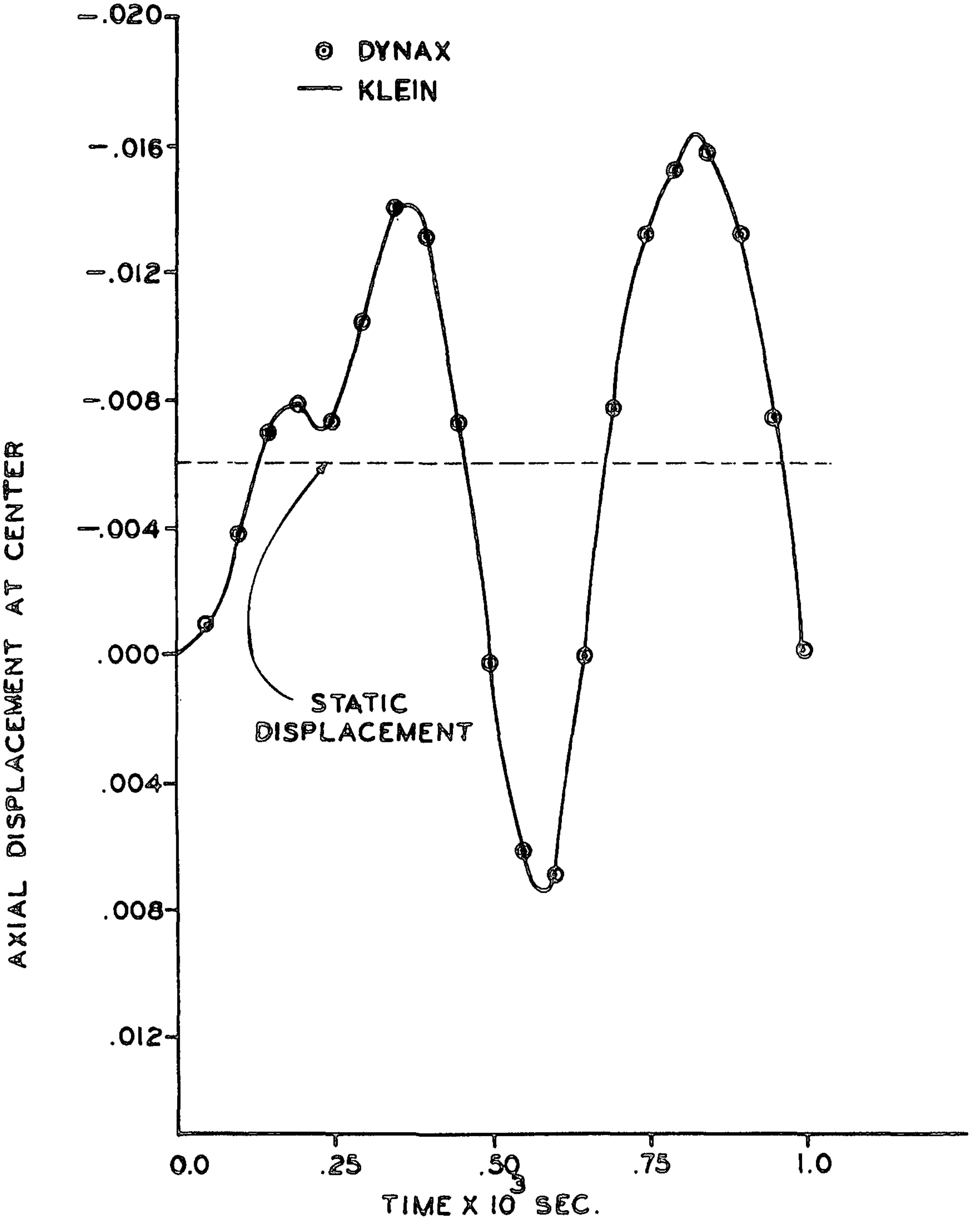
BENDING MOMENT VS. TIME
SUDDENLY APPLIED RING (LINE) LOAD



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-15

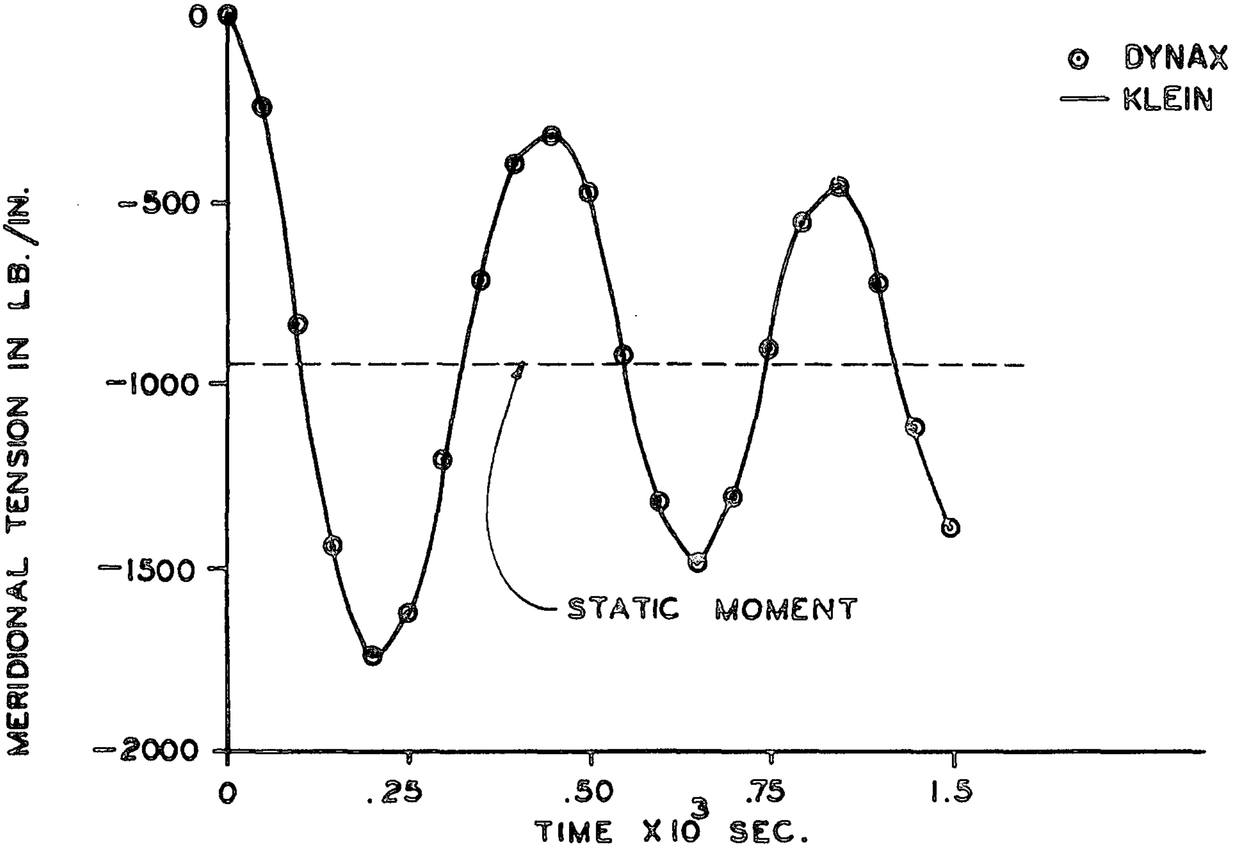
SPHERICAL CAP



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-16

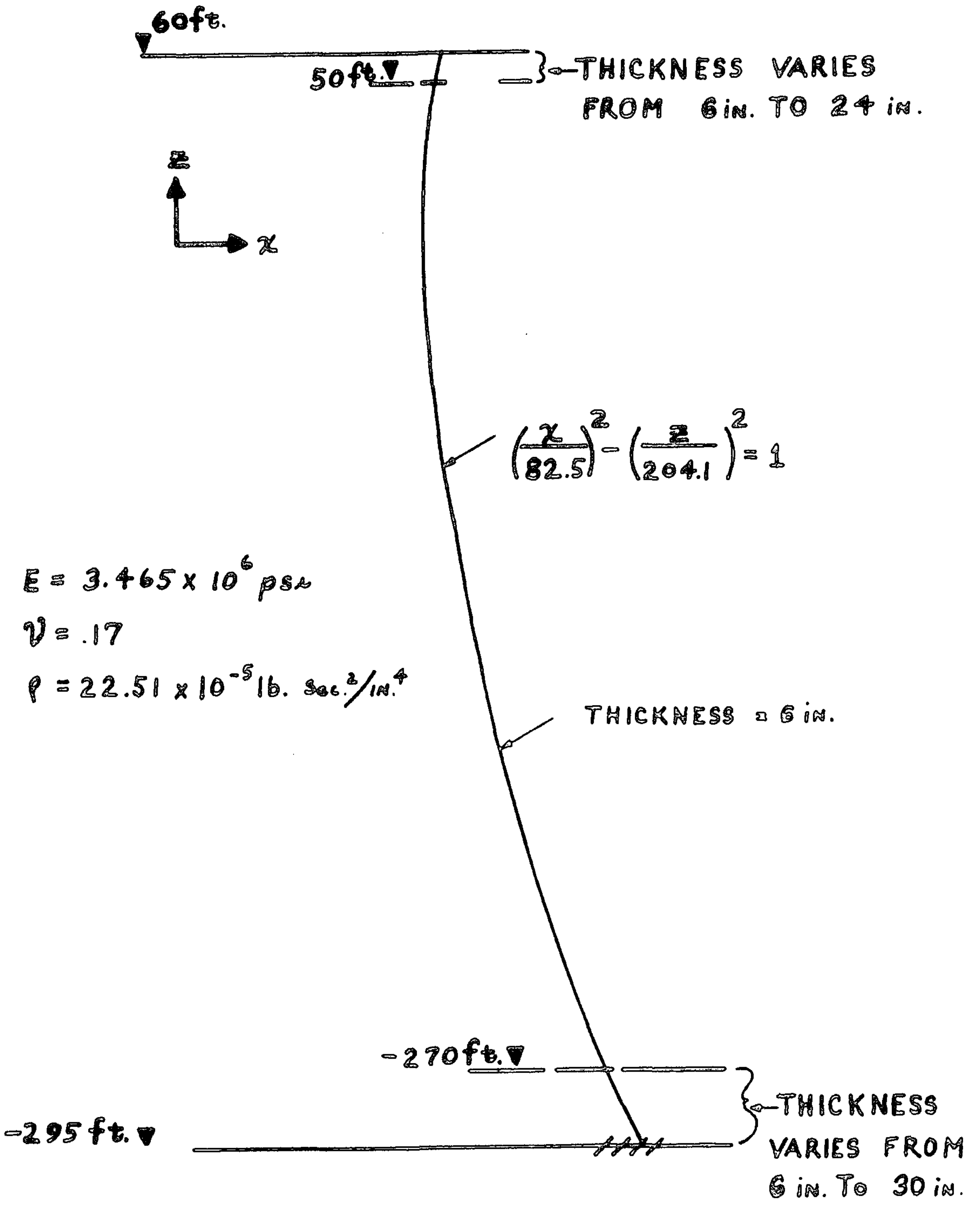
AXIAL DISPLACEMENT OF SPHERICAL CAP
UNDER DYNAMIC LOAD



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-17

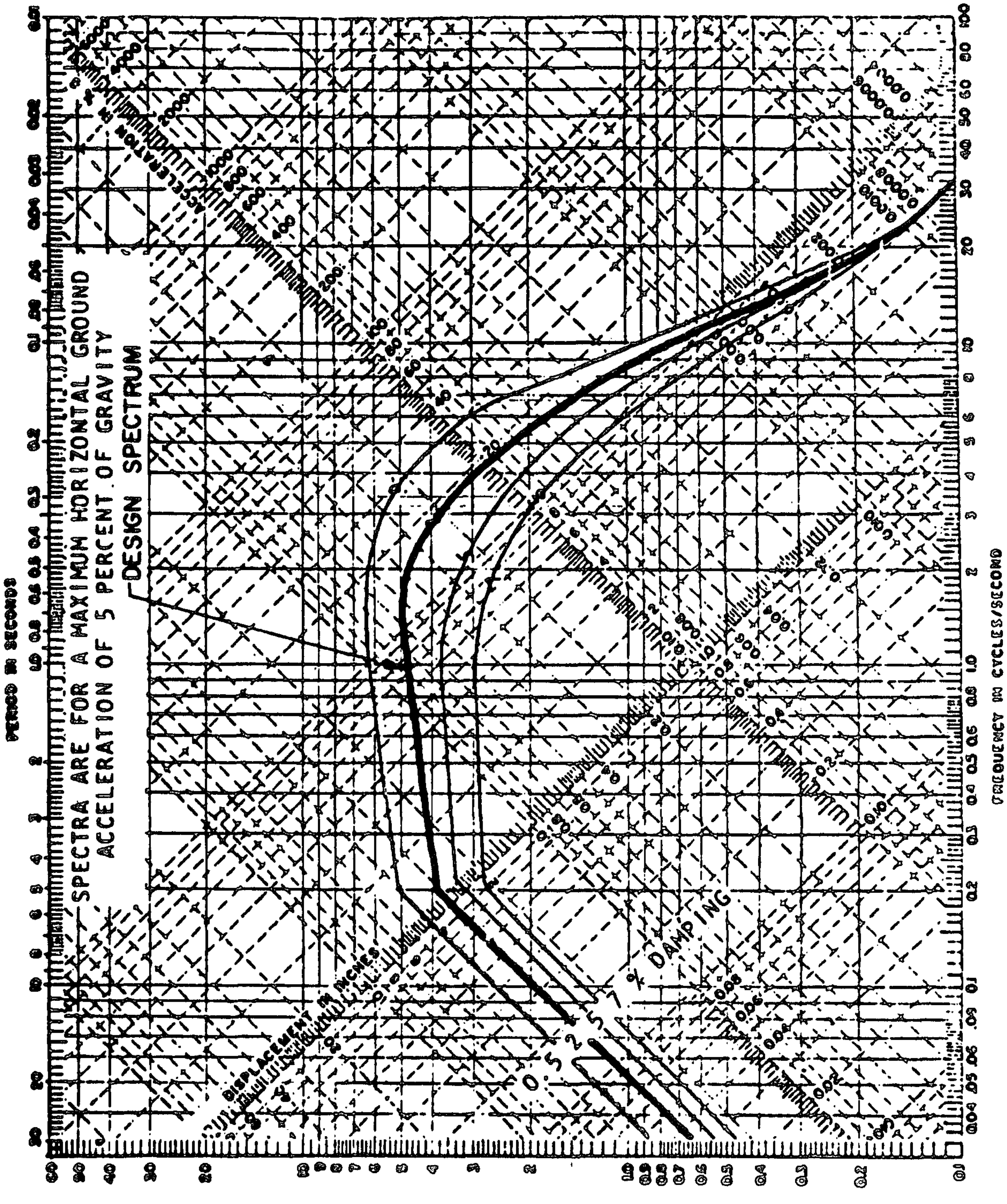
MERIDIONAL TENSION OF SPHERICAL CAP
UNDER DYNAMIC LOAD



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-18

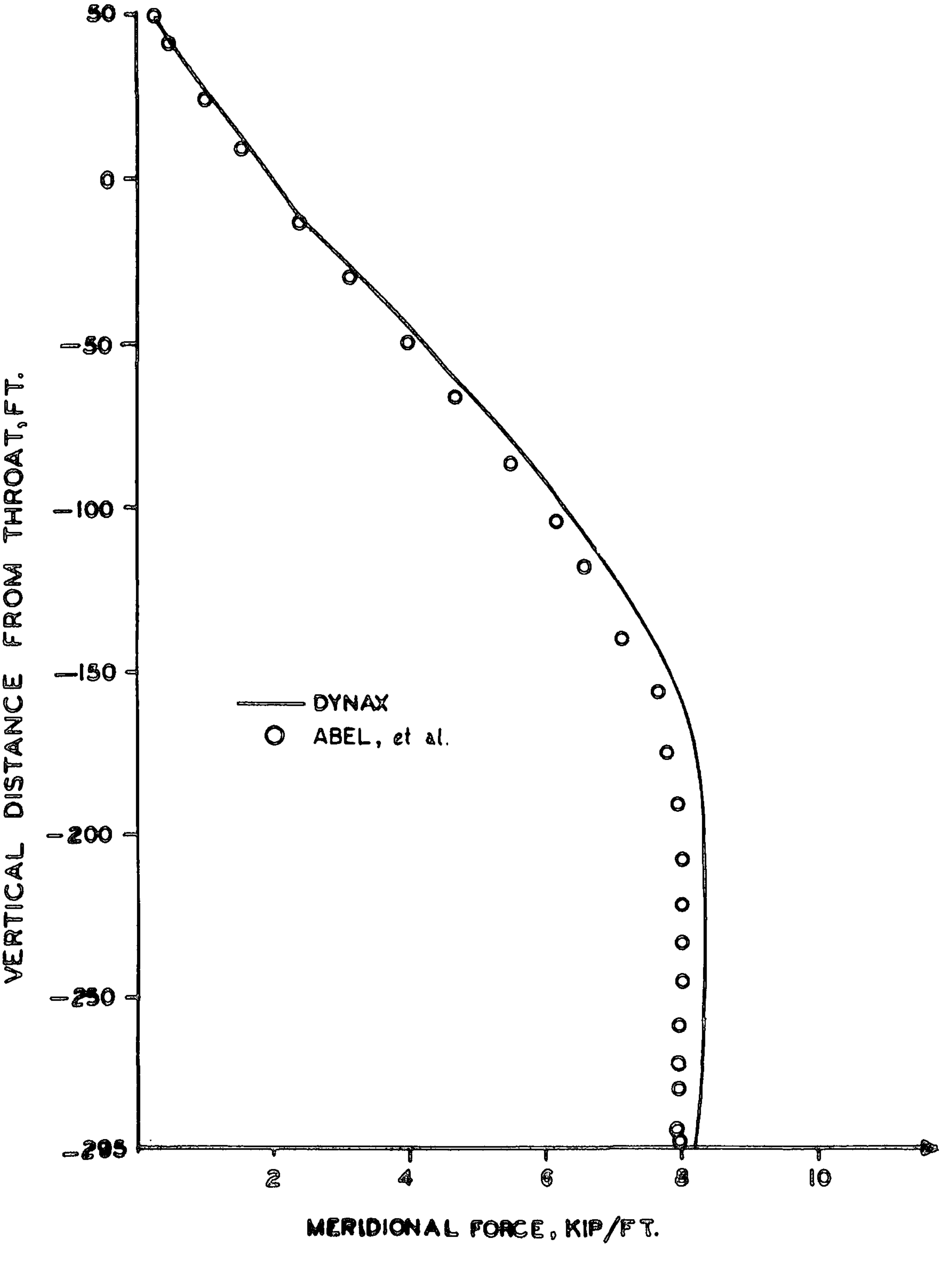
HYPERBOLIC COOLING TOWER



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-19

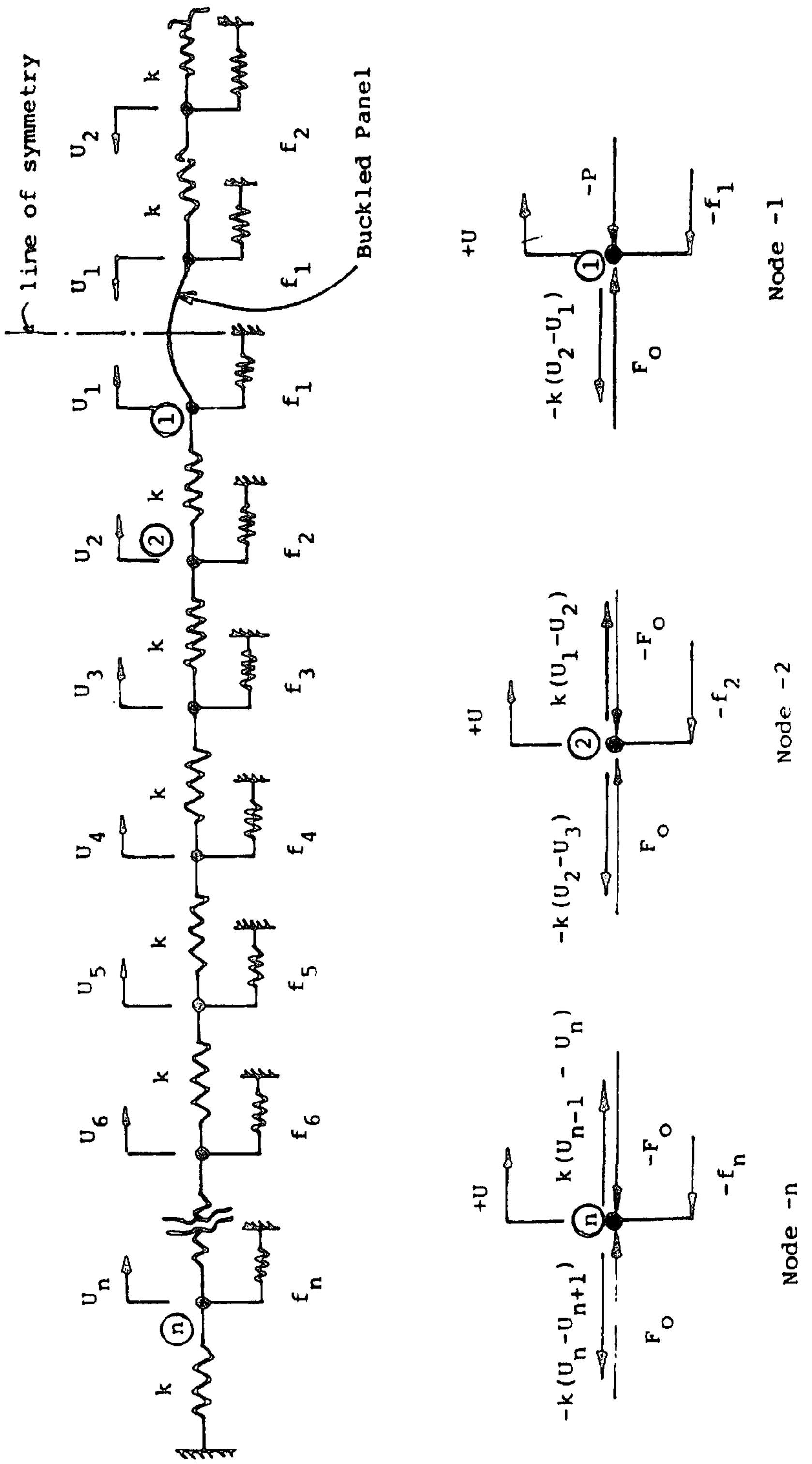
SPECTRUM OF DESIGN EARTHQUAKE



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-20

COOLING TOWER MERIDIONAL FORCE



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-21

IDEALIZED MODEL OF ANCHOR-PANEL SYSTEM
USED IN LAFD VALIDATION PROBLEM

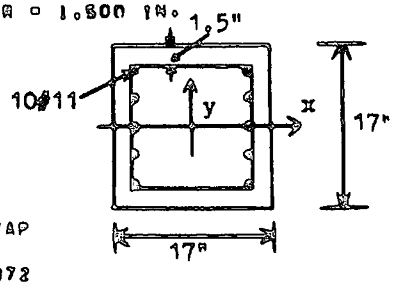
P.C.A. - U.S.D. OF R.C. COLUMNS

VALIDATION PROBLEM NO.1 - DESIGN OF A TIED COLUMN -COMPRESSION CONTROL

DESIGN OF
TIED COLUMN

B= 17.00 T= 17.00 FC= 3.000 FY=40.000 PHIC= .700 PHIB= .900

USE- 10 NO. 9 BARS. AST = 10.00 SQ.IN. = 3.47 PCT. COVER = 1.500 IN.



	ROW 1	ROW 2	ROW 3	ROW 4
NO. OF BARS	2	2	3	3
COVER	1.500	1.500	1.500	1.500

LOAD CASE	APPLIED FORCES AP	AMX	AMY	ULTIMATE CAPACITY UP	UMX	UMY	UP/AP
1	525.	0.	105.	563.	0.	113.	1.072
2	525.	75.	0.	603.	86.	0.	1.190

INTERACTION CONTROL POINTS REQUESTED

	PZ	PB	MB	MZ
X -AXIS	778.0	309.7	166.2	176.2
Y -AXIS	778.0	245.8	239.6	199.7
Z -AXIS	778.0	319.6	167.2	193.7

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-22

DESIGN OF TIED COLUMN
COMPRESSION CONTROLS

P.C.A. - U.S.D. OF R.C. COLUMNS

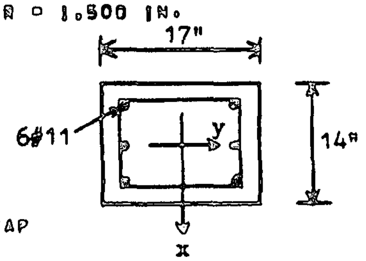
VALIDATION PROBLEM NO. 2 - DESIGN OF A TIED COLUMN - TENSION CONTROLS

DESIGN OF
TIED COLUMN

B= 14.00 T= 20.00 FC= 4.500 FY=50.000 PHIC= .700 PHIB= .900

USE- 6 NO.11 BARS. AST = 9.36 SQ.IN. = 3.35 PCT. COVER = 1.500 IN.

	ROW 1	ROW 2	ROW 3	ROW 4
NO. OF BARS	3	3	0	0
COVER	1.500	1.500	1.500	1.500



LOAD CASE	APPLIED FORCES AP	AMX	AMY	ULTIMATE CAPACITY UP	UMX	UMY	UP/AP
1	115.	279.	0.	122.	295.	0.	1.057
2	115.	0.	14.	801.	0.	99.	6.966

INTERACTION CONTROL POINTS REQUESTED

	PZ	PA	MO	MZ
X -AXIS	1052.2	317.9	353.8	282.0
Y -AXIS	1052.2	319.4	187.2	180.3
Z -AXIS	1052.2	310.9	231.3	254.0

FIGURE D-23

DESIGN OF TIED COLUMN
TENSION CONTROLS

P.C.A. - U.S.D. OF R.C. COLUMNS

VALIDATION PROBLEM NO.3 - DESIGN OF A TIED COLUMN-BIAXIAL BENDING

DESIGN OF
TIED COLUMN

B= 28.00 T= 28.00 FC= 5.000 FY=60.000 PHIC= .700 PHIB= .900

USE= 12 NO.11 BARS. AST = 18.72 SQ.IN. = 2.39 PCT. COVER = 1.500 IN.

	ROW 1	ROW 2	ROW 3	ROW 4
NO. OF BARS	4	4	2	2
COVER	1.500	1.500	1.500	1.500

LOAD	APPLIED FORCES			ULTIMATE CAPACITY			
CASE	AP	AMX	AMY	UP	UMX	UMY	UP/AP
1	1330.	790.	0.	1626.	966.	0.	1.223
2	1330.	0.	394.	2216.	0.	655.	1.666
3	1330.	790.	394.	1388.	829.	411.	1.044

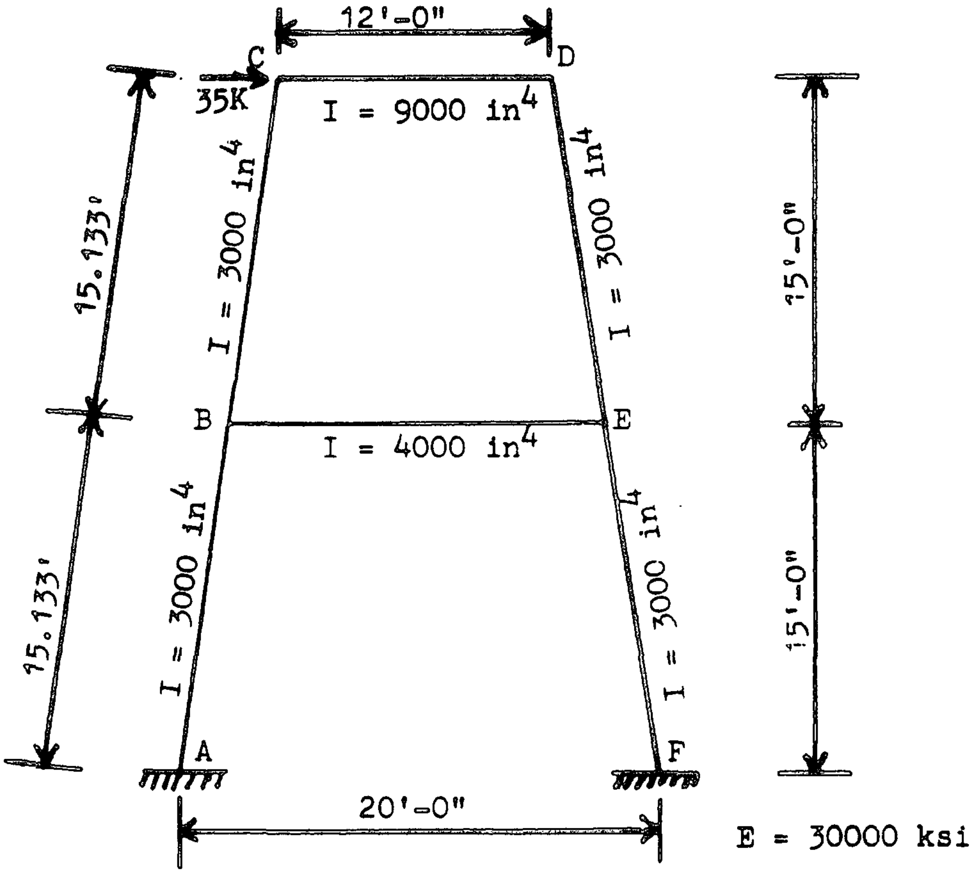
INTERACTION CONTROL POINTS REQUESTED

	PZ	PB	MB	MZ
X -AXIS	3062.9	983.0	1167.4	999.1
Y -AXIS	3062.9	983.0	1167.4	999.1
Z -AXIS	3062.9	910.2	949.7	947.4

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-24

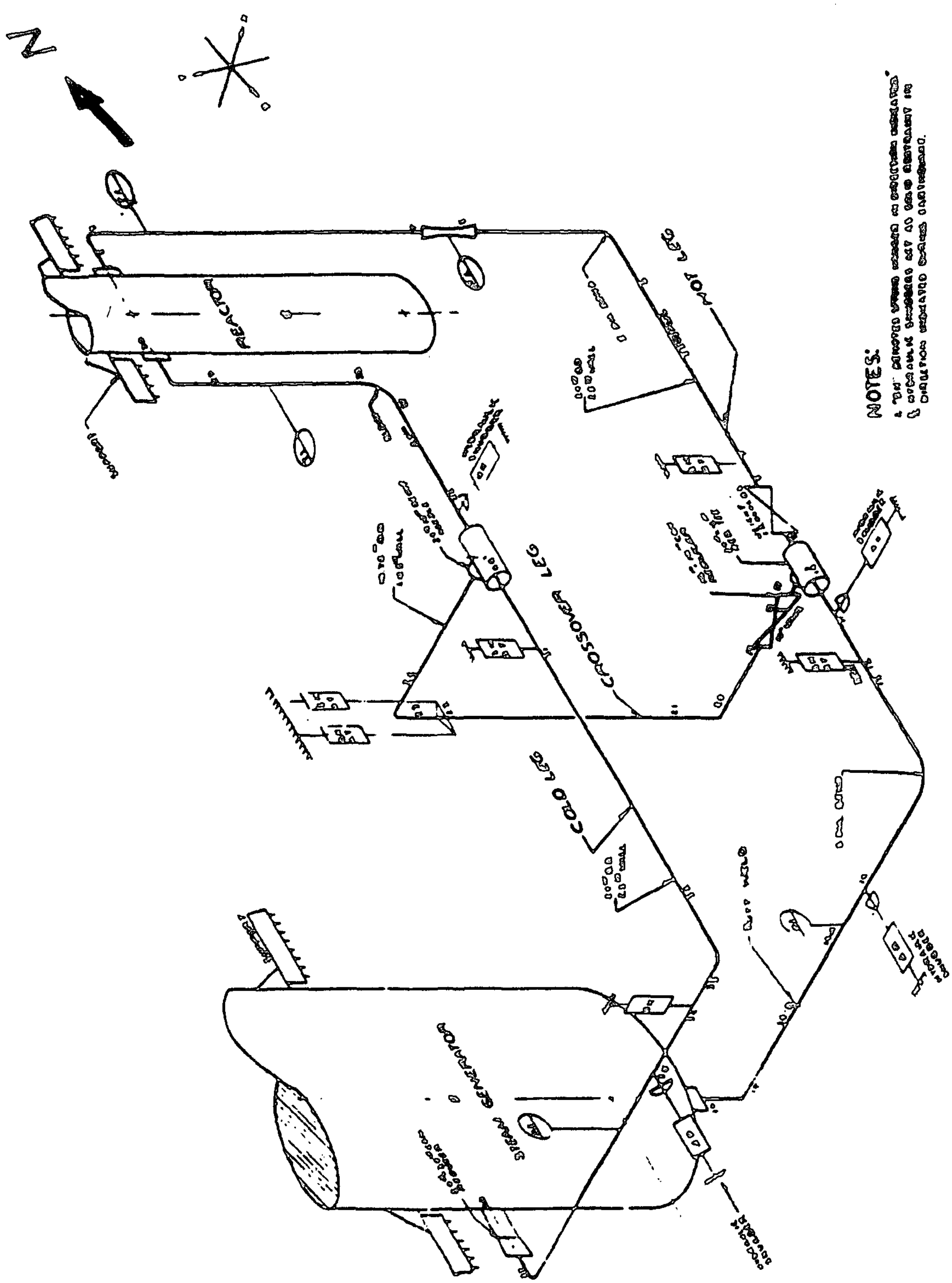
DESIGN OF TIED COLUMN
BIAXIAL BENDING



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-25

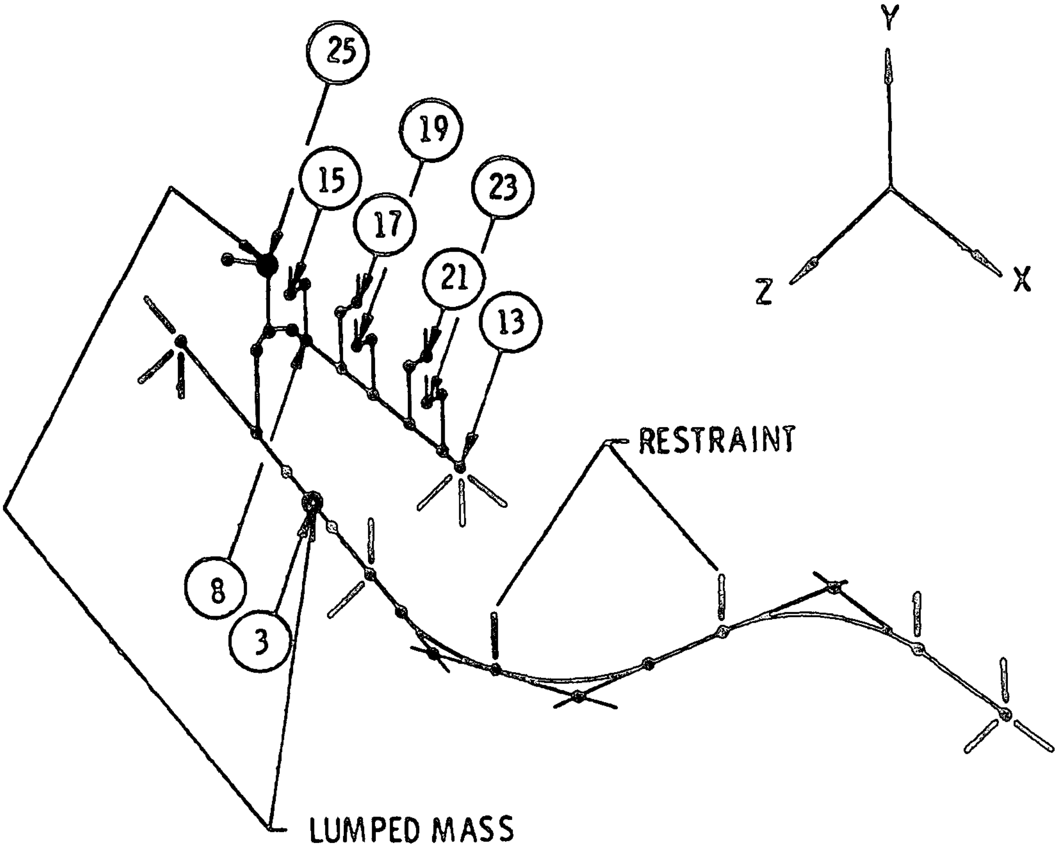
EXAMPLE FRAME FOR
PIPSYS STATIC ANALYSIS



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-26

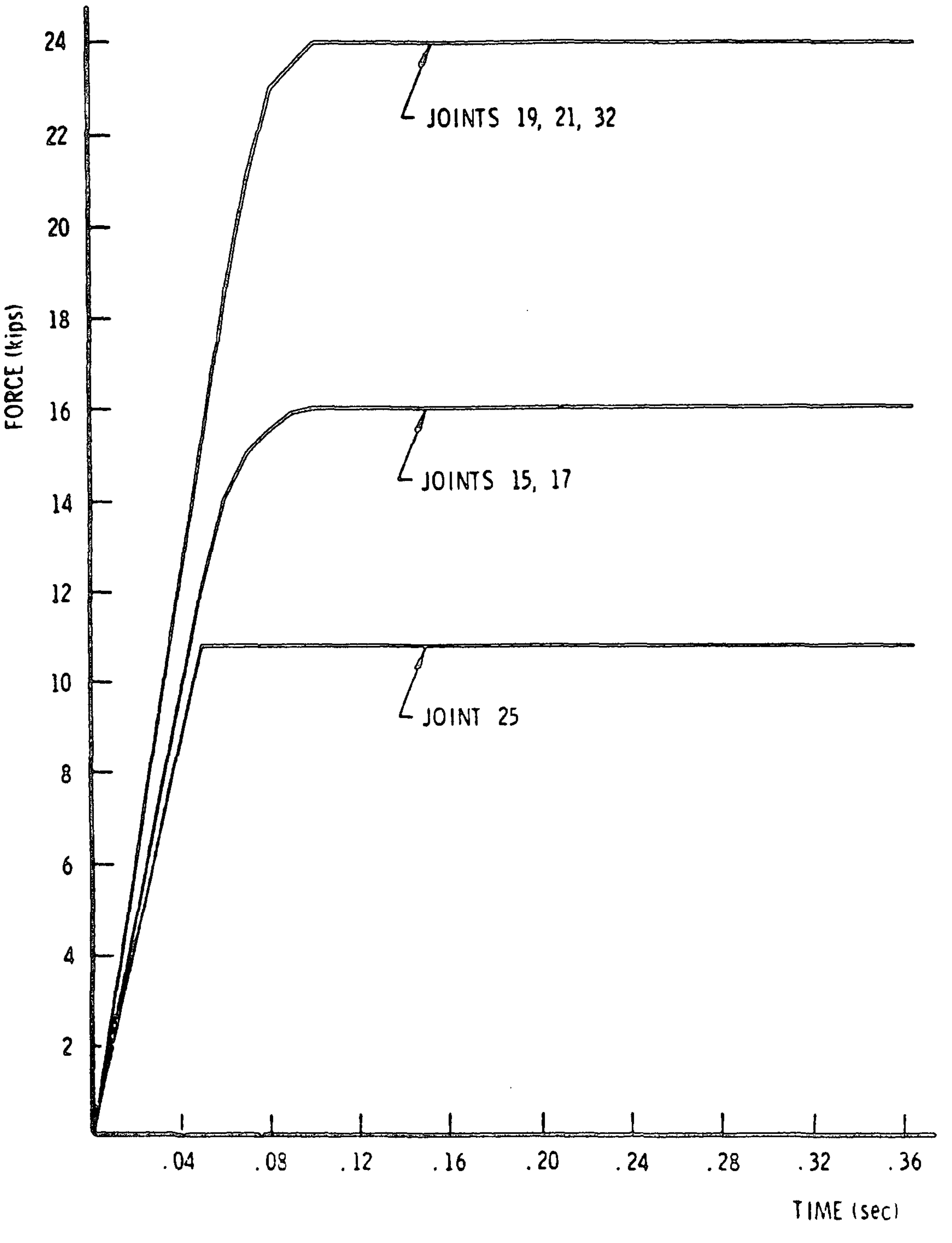
PIPING SYSTEM FOR COMBINED
STRESS ANALYSIS (PIPSYS)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-27

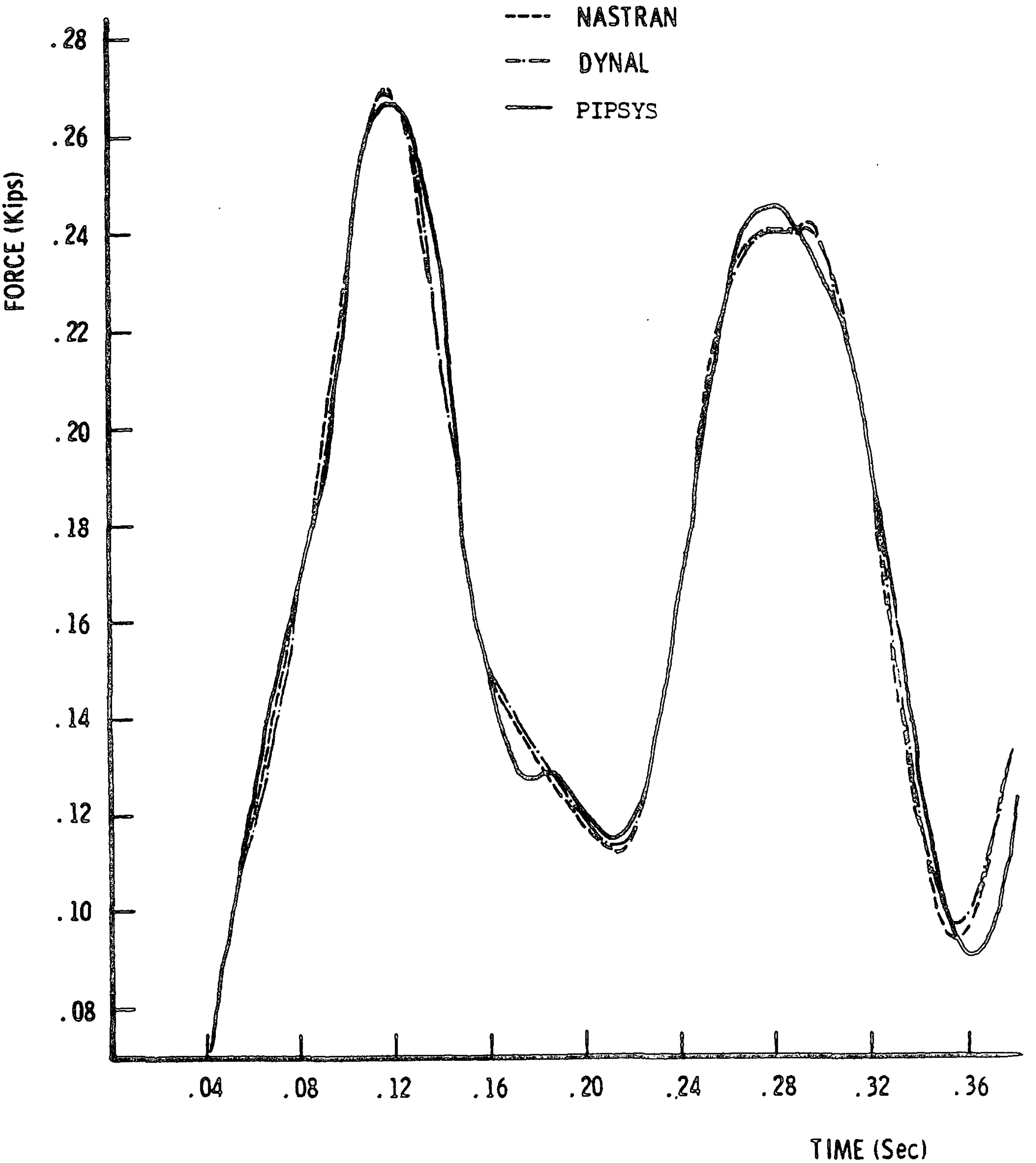
STRUCTURAL MODEL OF PIPING SYSTEM (PIPSYS)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-28

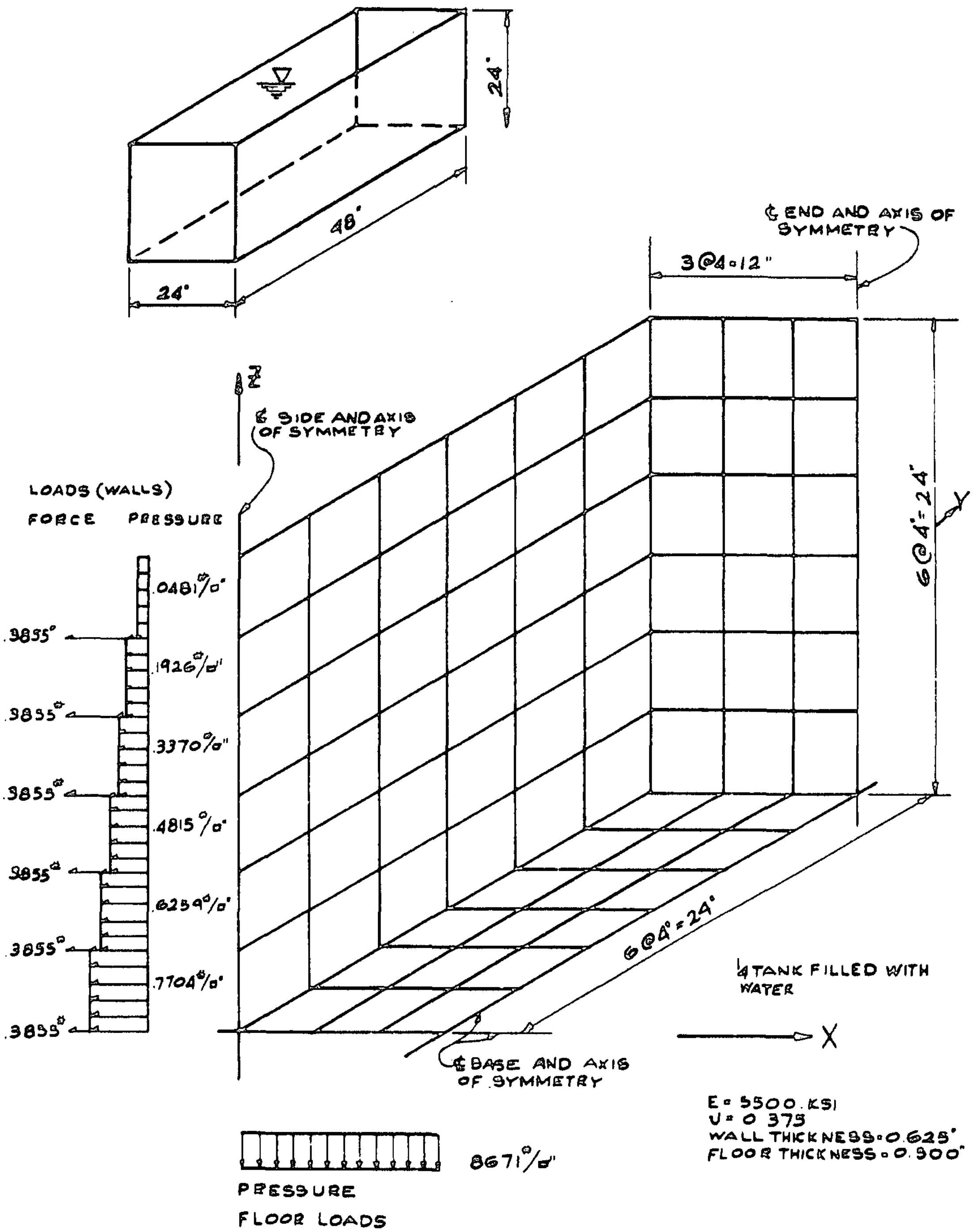
LOAD TIME HISTORY (PIPSYS)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-29

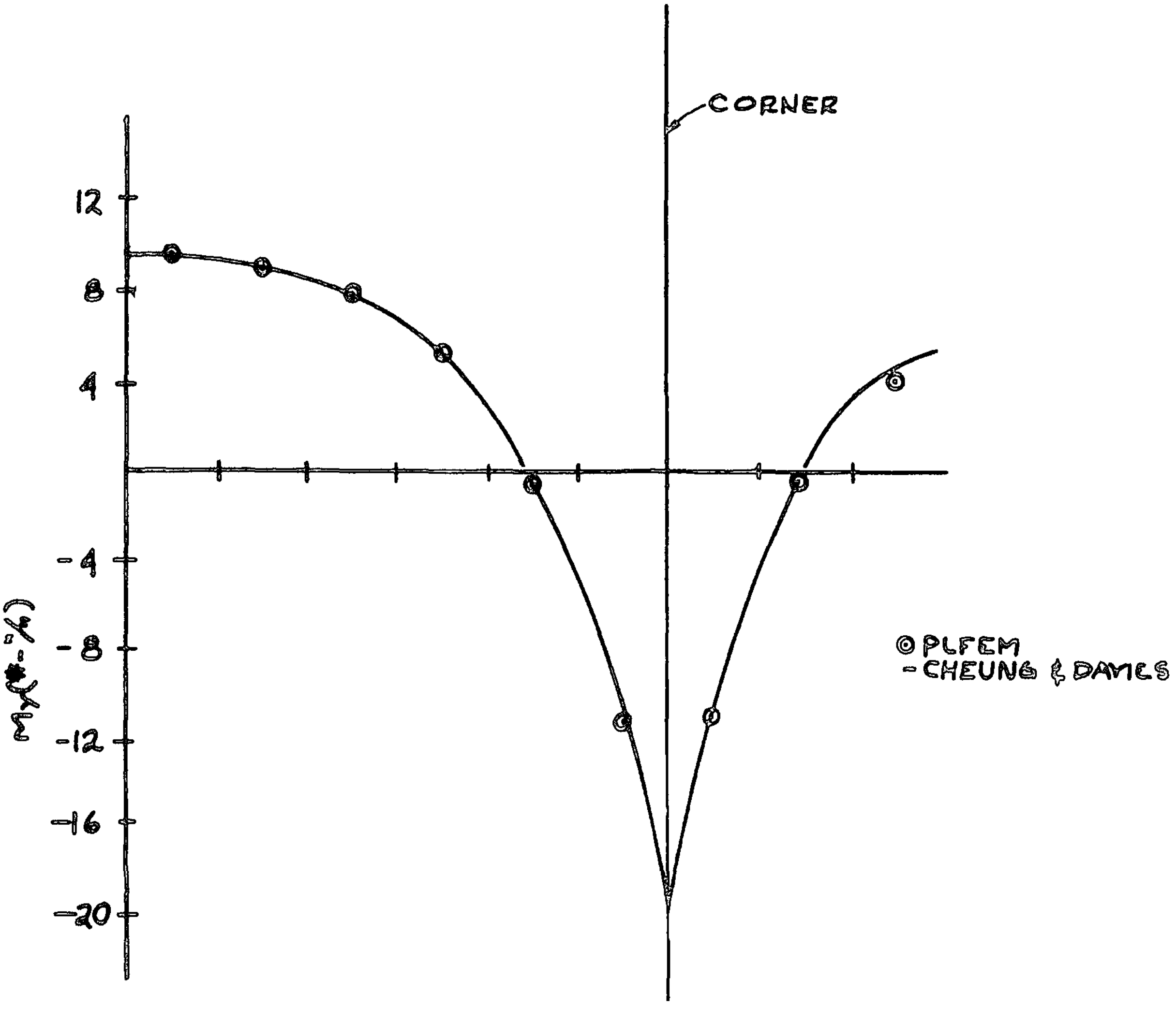
FORCE VS. TIME JOINT 8 Z DIRECTION
(PIPSYS)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-30

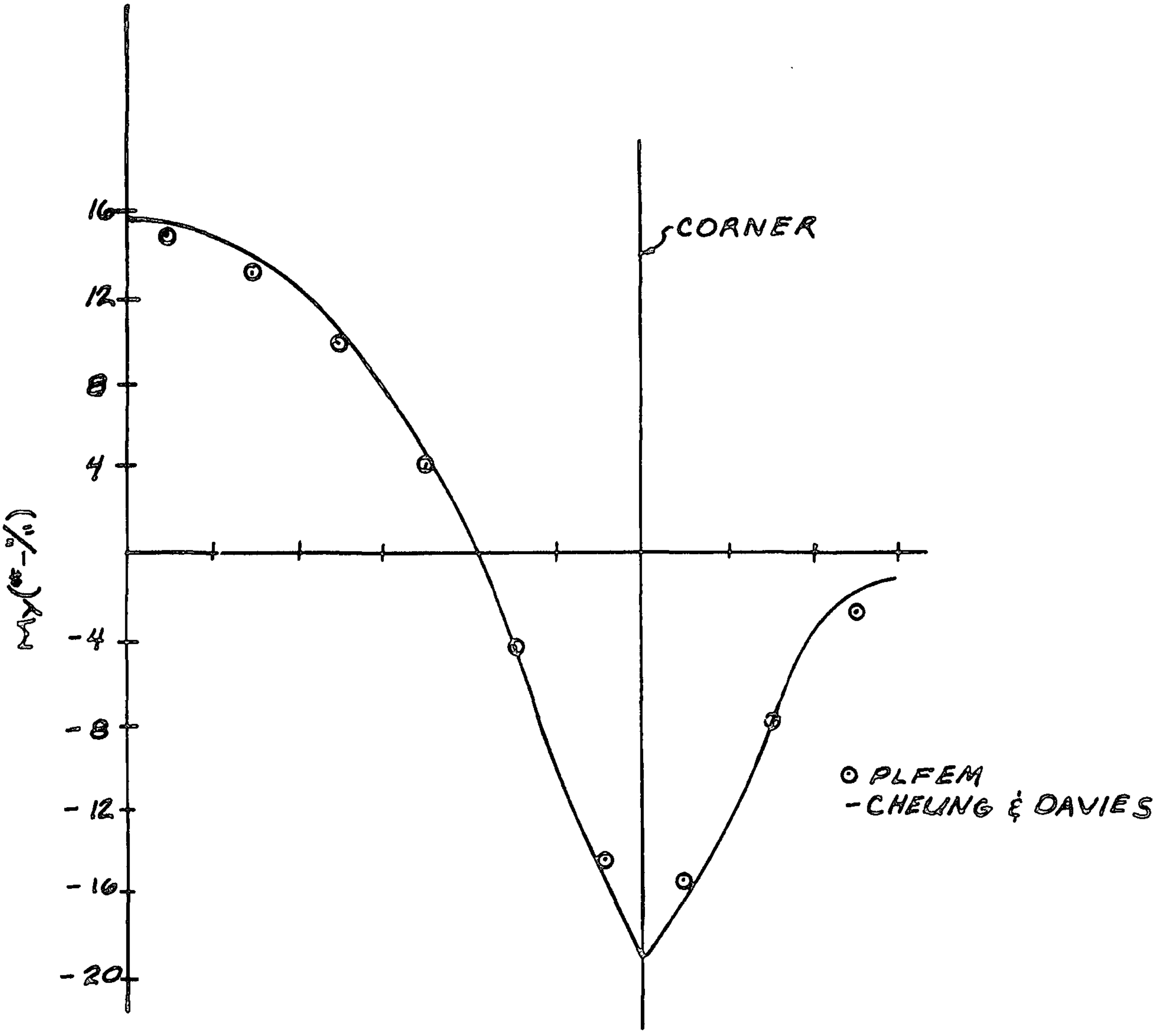
RECTANGULAR TANK FILLED WITH WATER



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-31

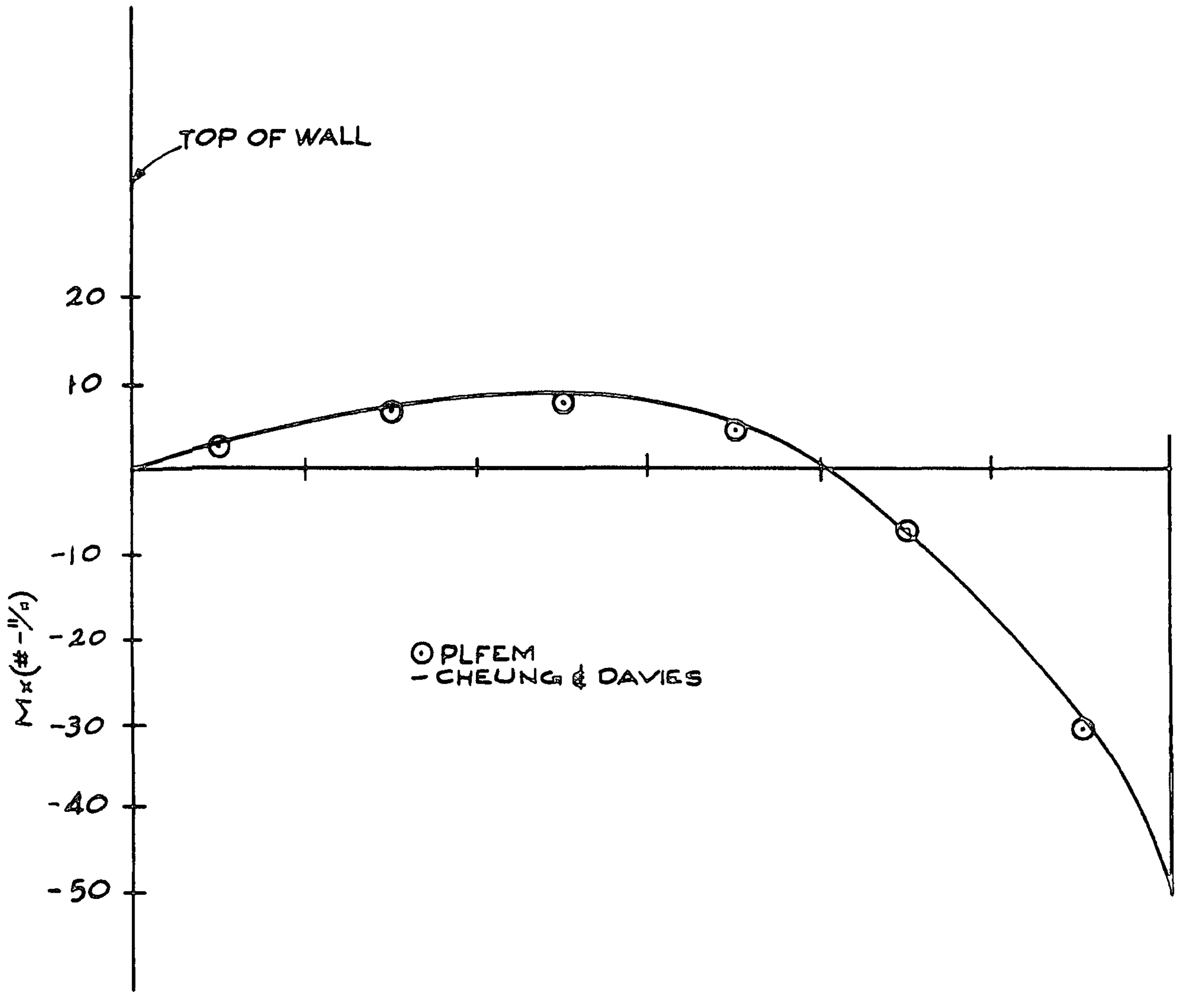
MOMENT OF M_y AT HORIZONTAL CROSS SECTION OF WALLS



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-32

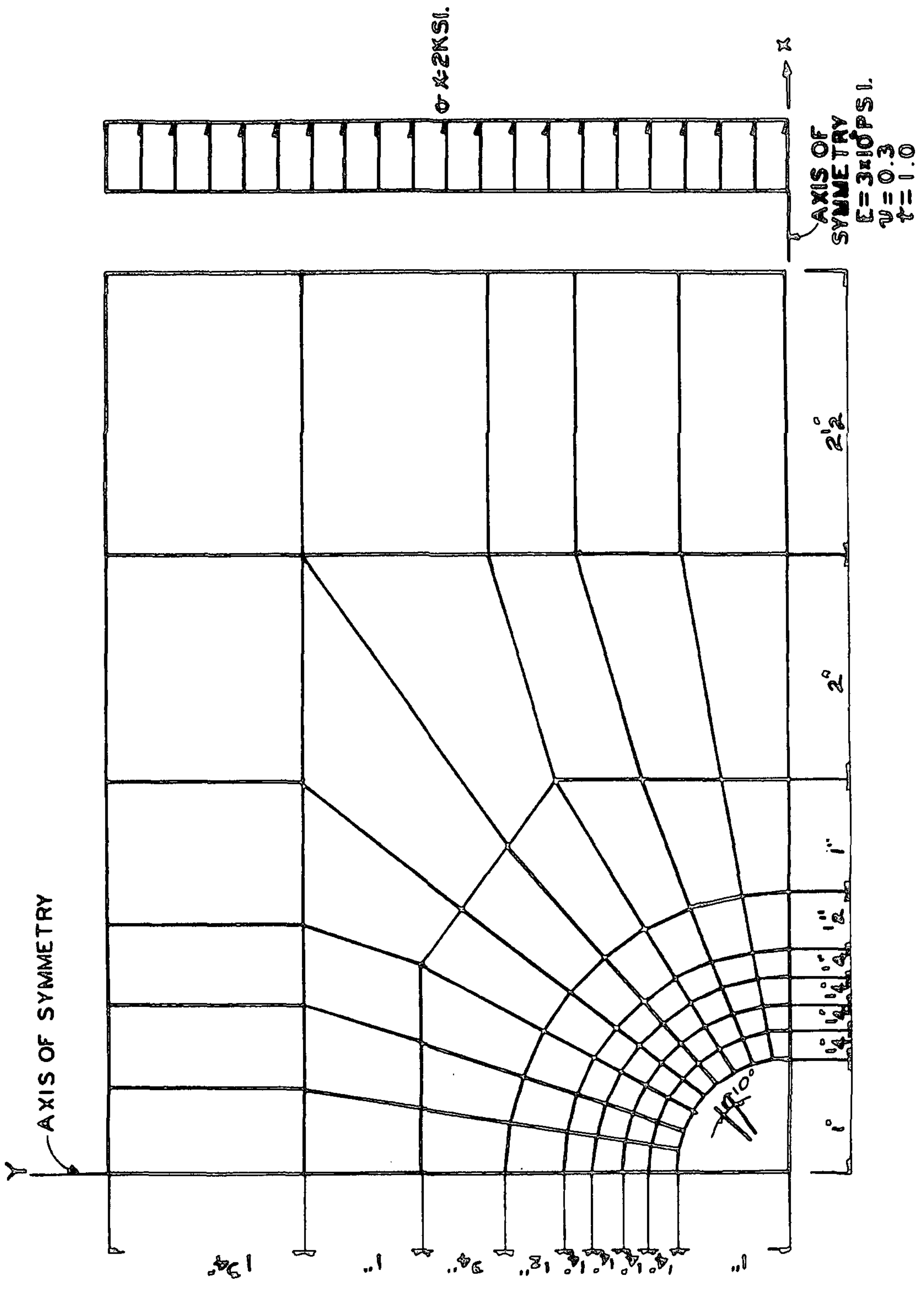
MOMENT M_y AT TOP OF WALL



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-33

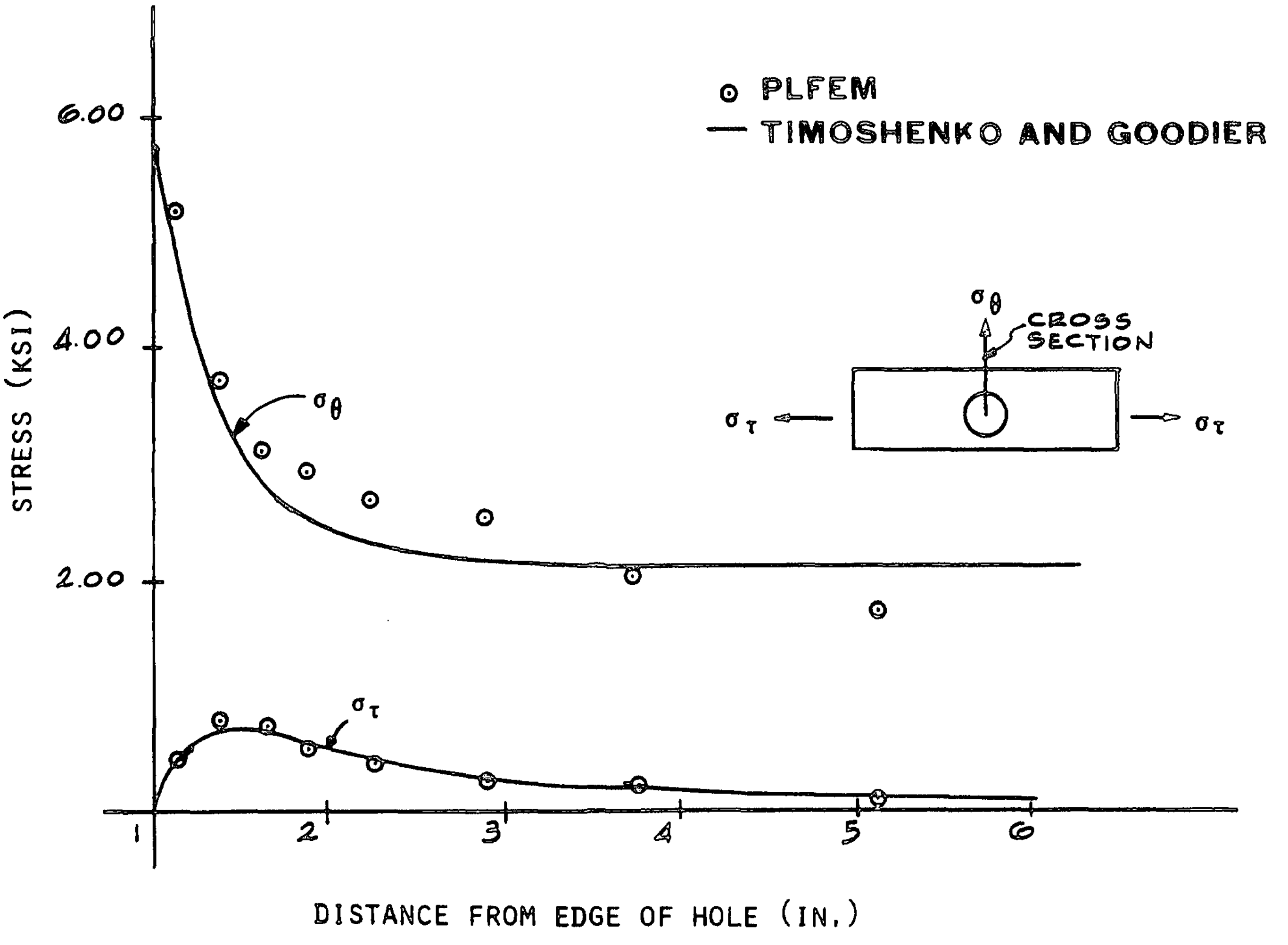
MOMENT M_x ALONG CROSS SECTION
OF LONG WALL



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-34

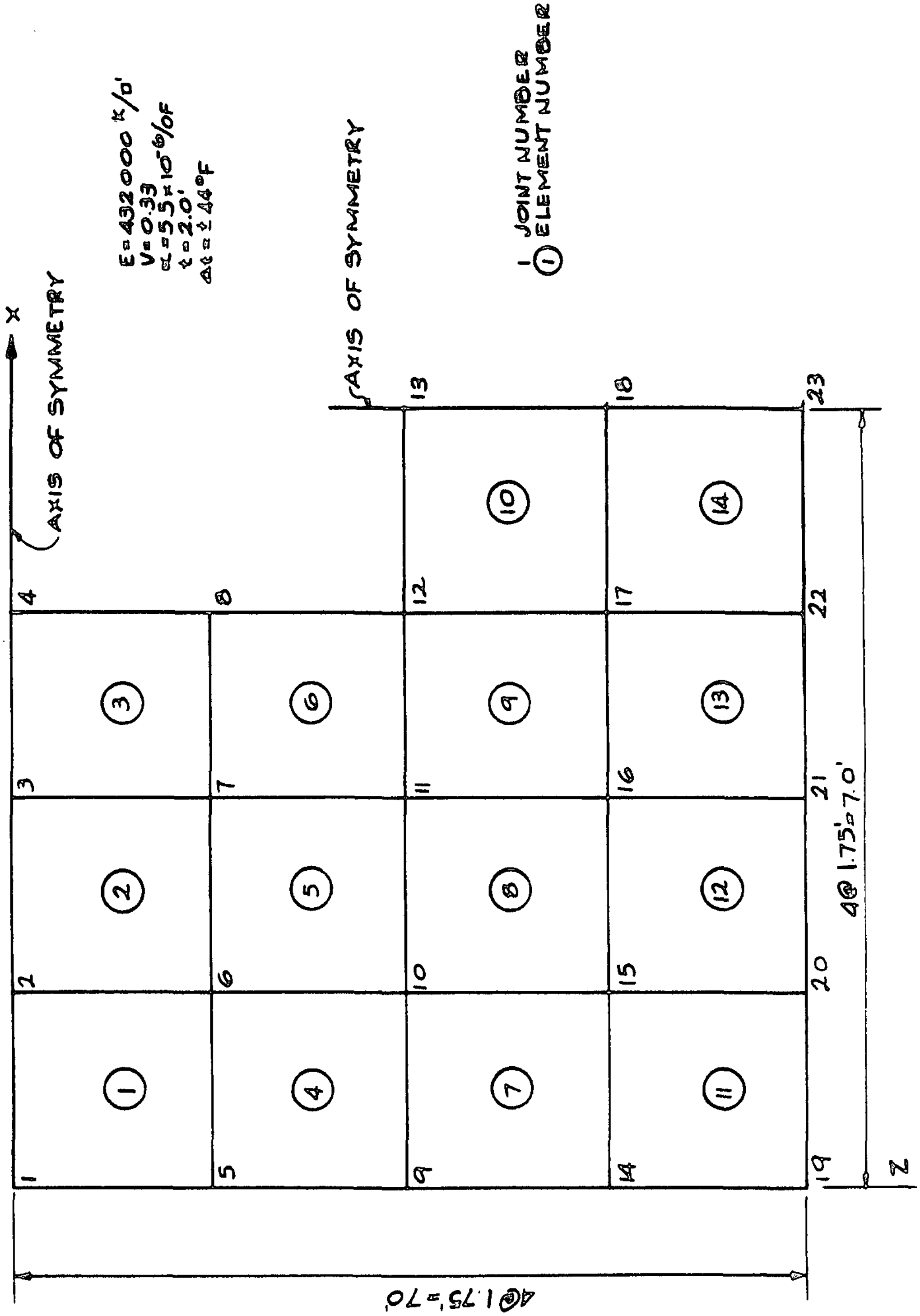
PLATE WITH CIRCULAR HOLE
UNDER UNIFORM TENSION



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-35

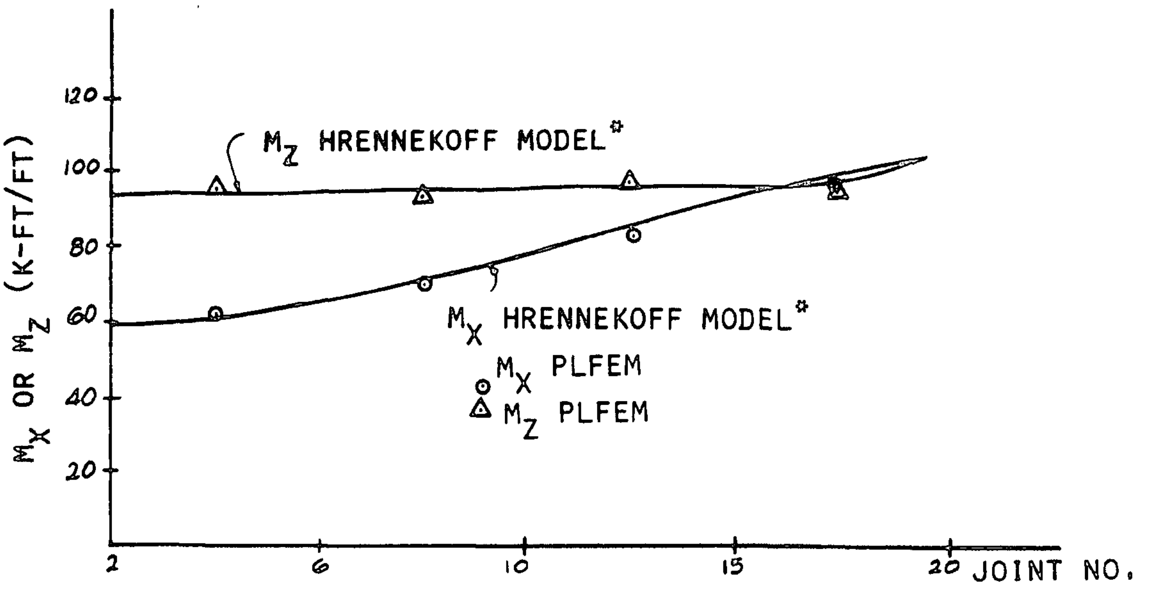
STRESSES IN PLATE WITH CIRCULAR HOLE
UNDER UNIFORM TENSION



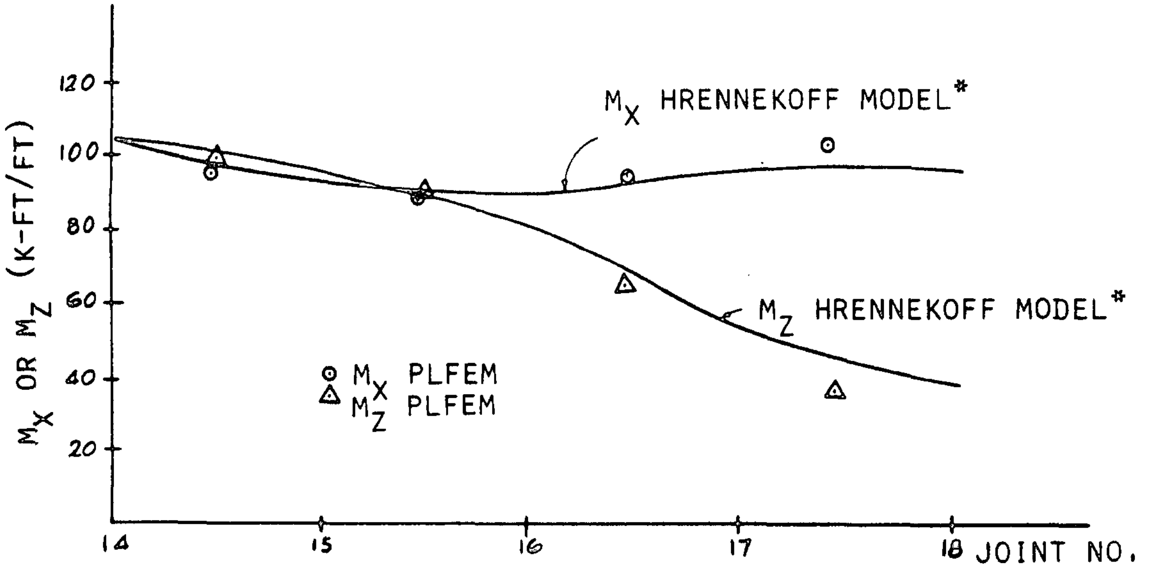
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-36

SQUARE PLATE WITH RECTANGULAR HOLE
SUBJECTED TO TEMPERATURE VARIATION



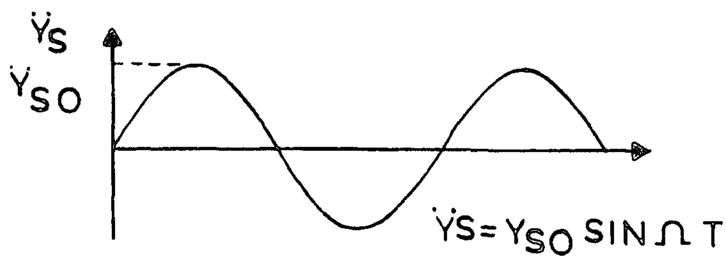
M_X AND M_Z ALONG THE 2-20 LINE



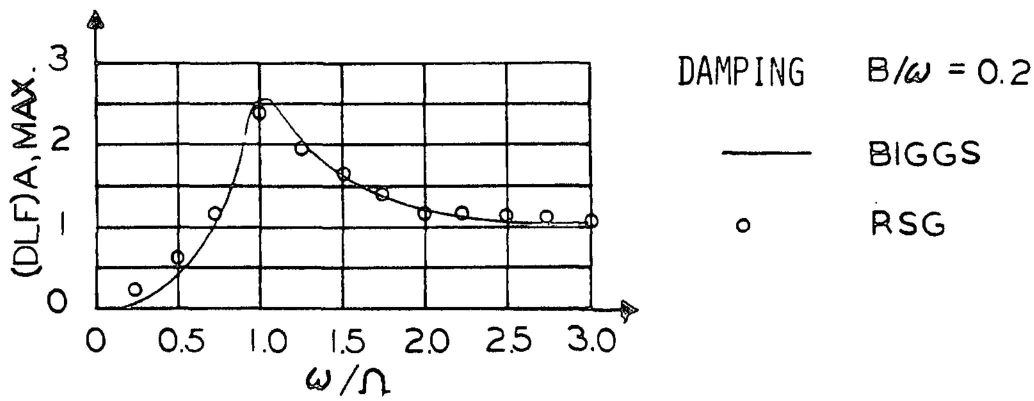
M_X AND M_Z ALONG THE 14-18 LINE

* HRENNEKOFF MODEL BASED ON A FRAMEWORK ELEMENT 0.875' SQUARE

BYRON/BRAIDWOOD STATIONS UPDATED FINAL SAFETY ANALYSIS REPORT
FIGURE D-37 MOMENTS IN PLATE DUE TO TEMPERATURE VARIATION



ACCELERATION TIME HISTORY

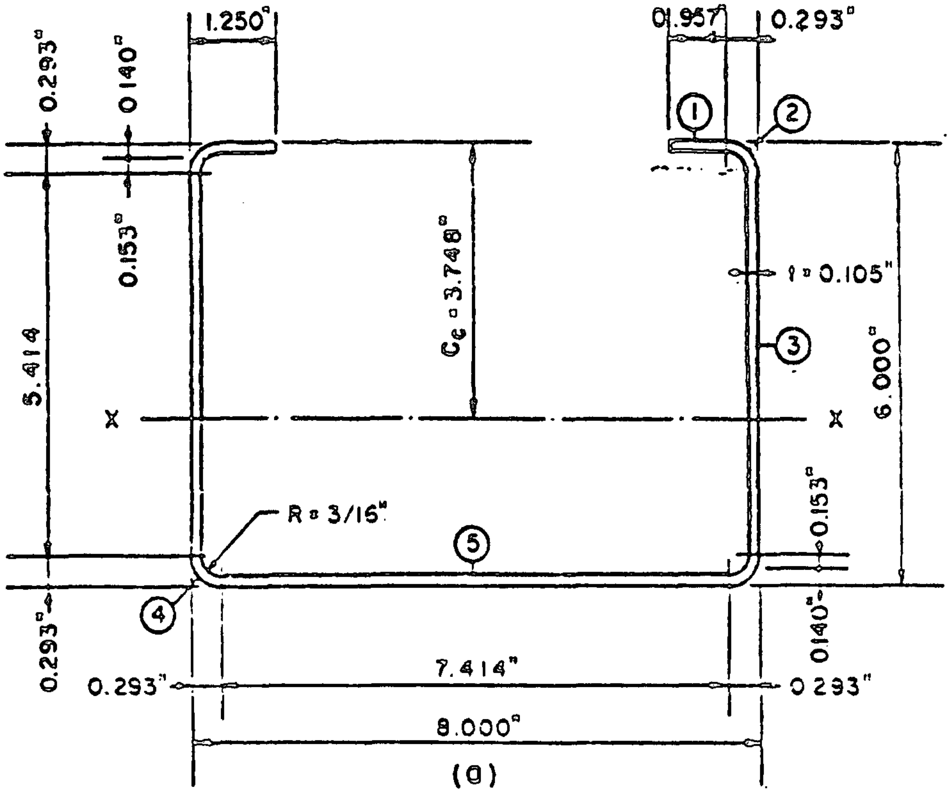


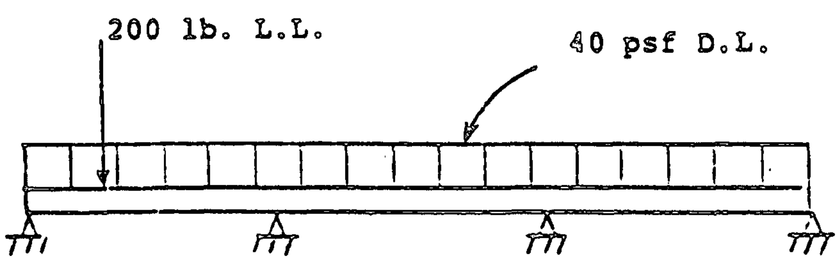
RESPONSE SPECTRUM

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-38

VALIDATION FOR A ONE-DEGREE-OF-FREEDOM DAMPED SYSTEM (RSG)





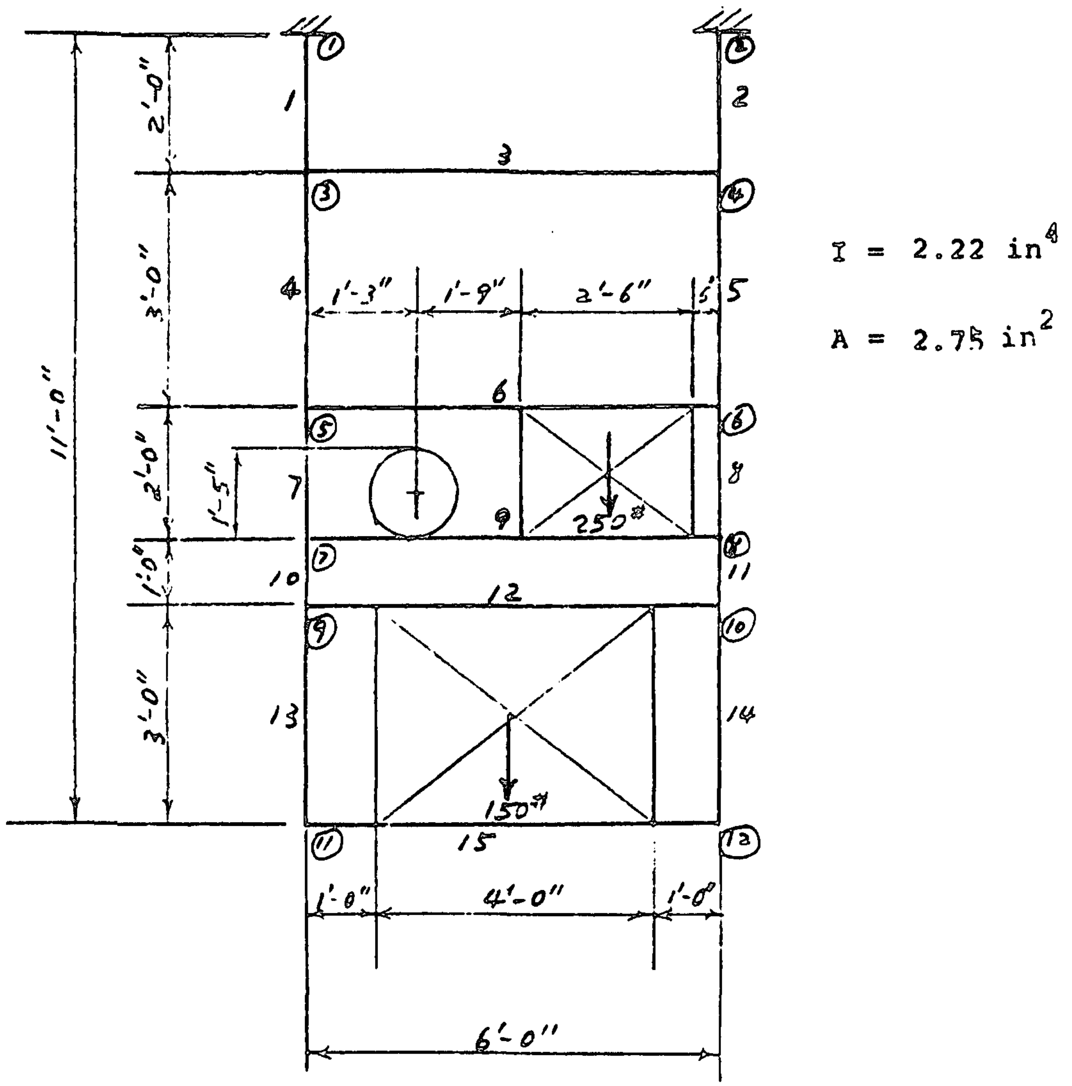
vertical seismic design load = 1.5g

horizontal seismic design load = 4.5g

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-39

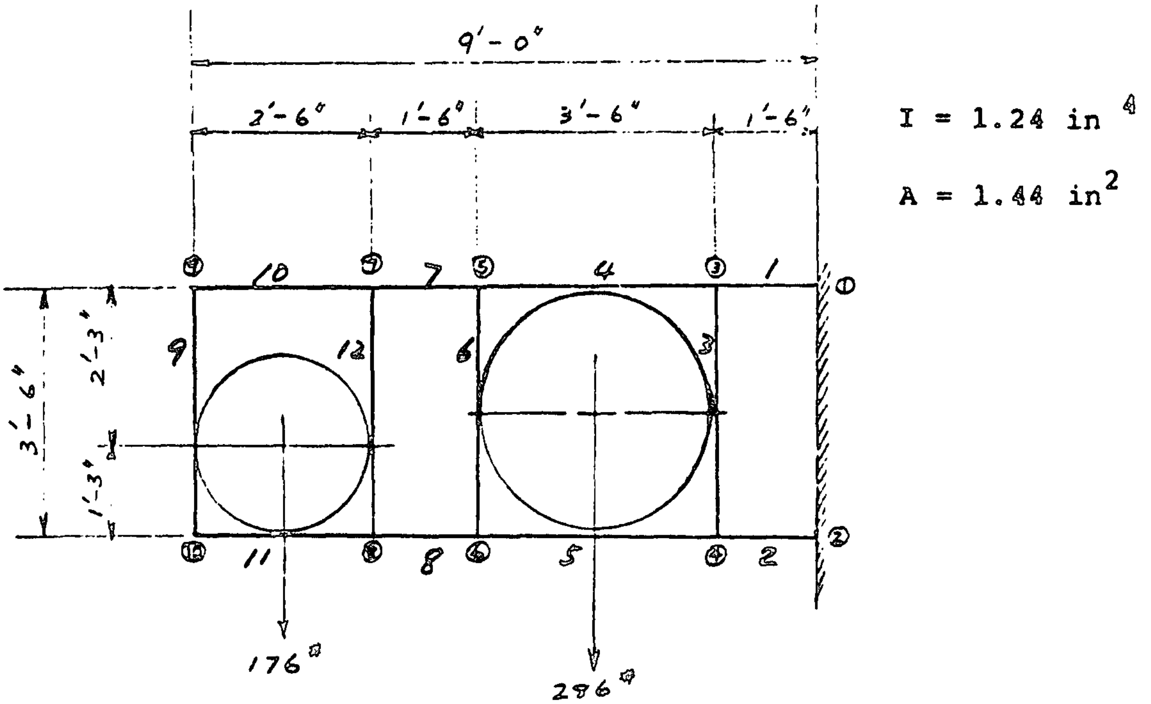
CABLE TRAY MODEL FOR SEISHANG PROGRAM



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-40

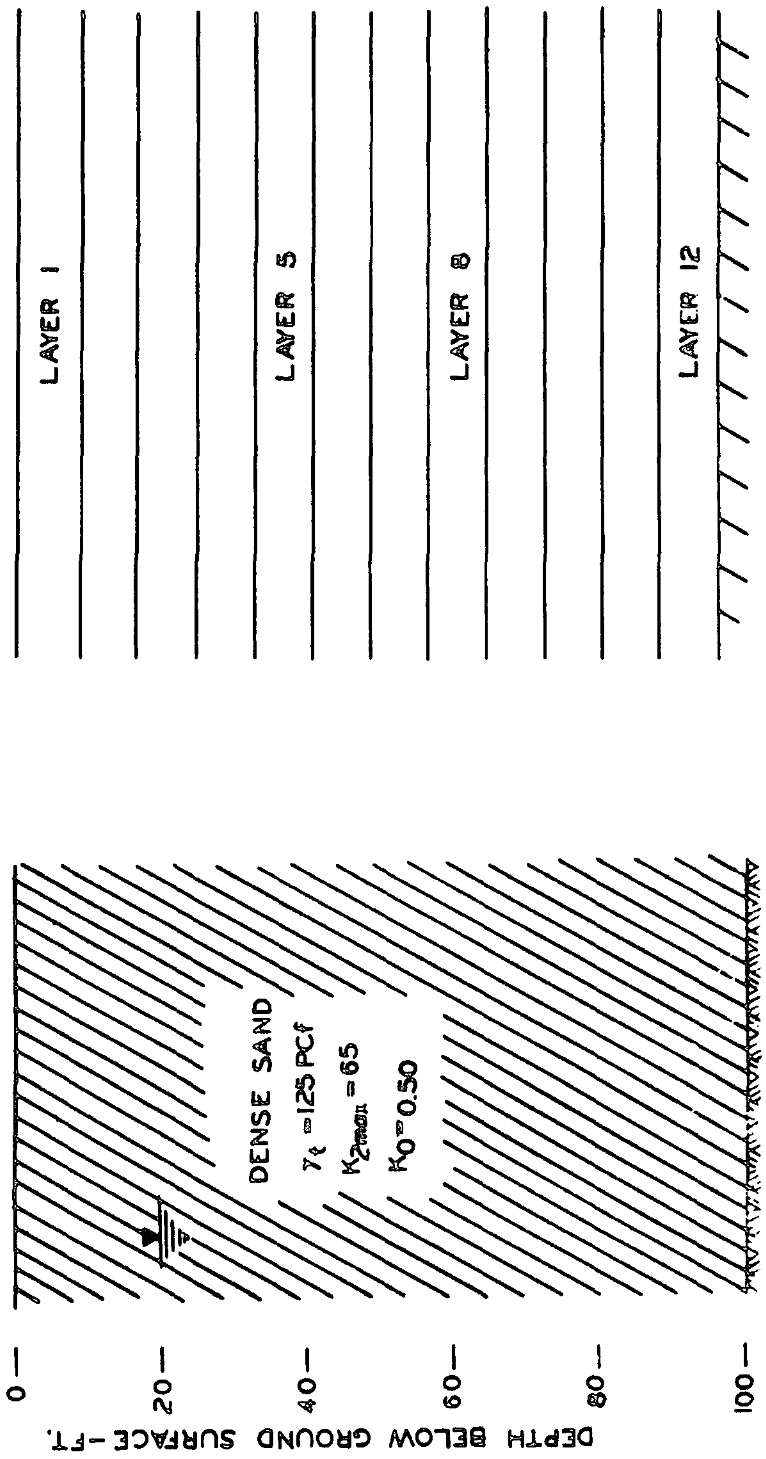
CEILING MOUNTED SUPPORT MODEL
FOR SEISHANG PROGRAM



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-41

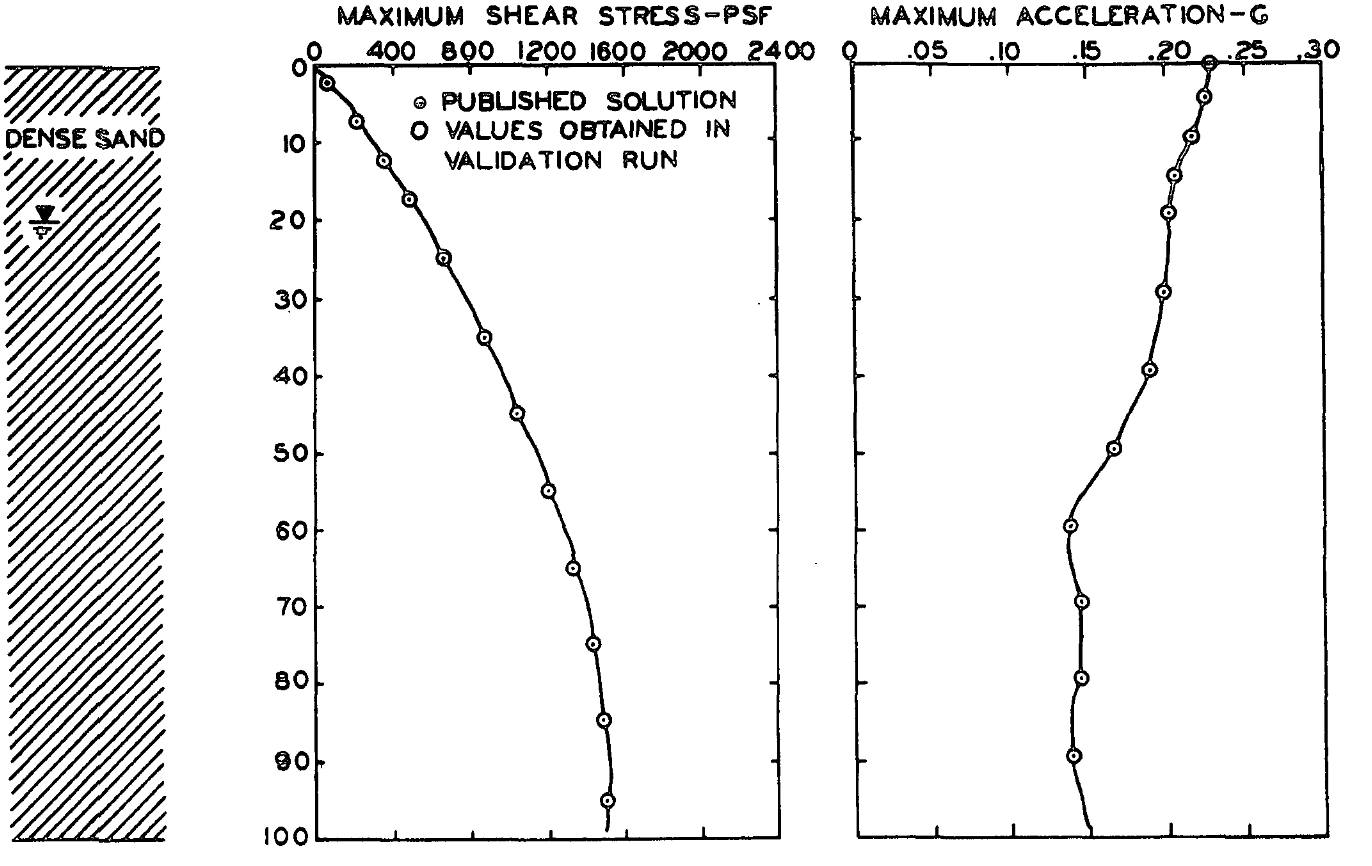
WALL MOUNTED SUPPORT MODEL
FOR SEISHANG PROGRAM



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-42

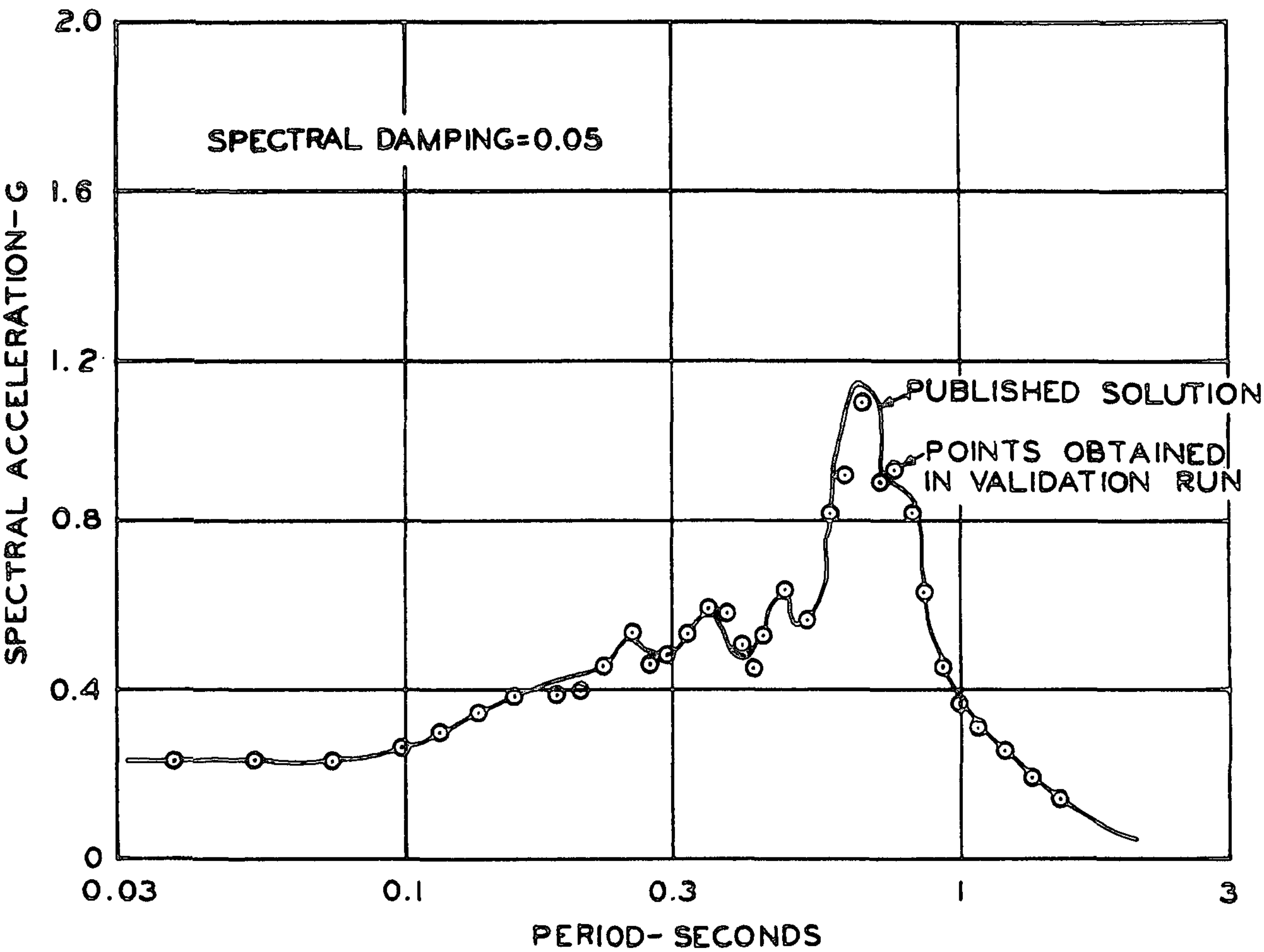
SOIL PROFILE AND LAYERED REPRESENTATION USED FOR SAMPLE PROBLEM



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-43

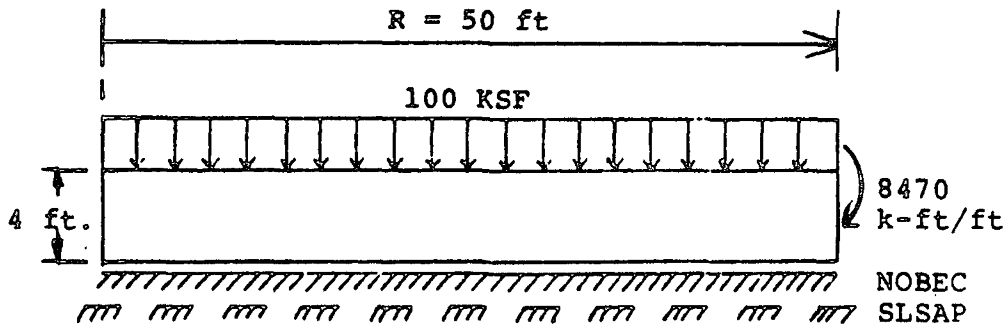
COMPARISON OF SHEAR STRESSES AND ACCELERATIONS



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-44

COMPARISON OF SPECTRAL VALUES
FOR SURFACE MOTIONS



$E = 593141.8$ kips/ft^2

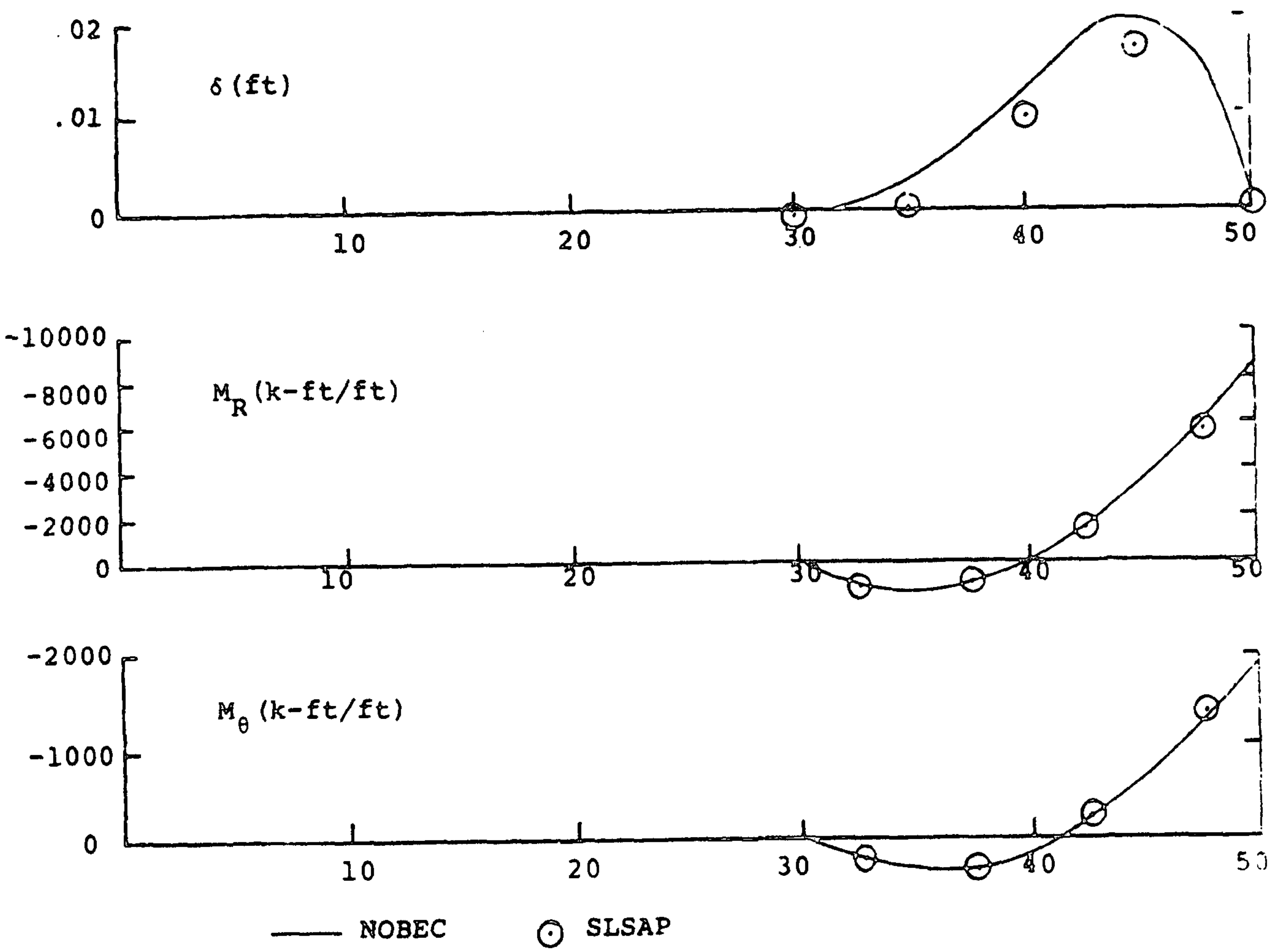
$\nu = 0.205$

$\rho = 0.15$ kips sec^2/ft^4

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-45

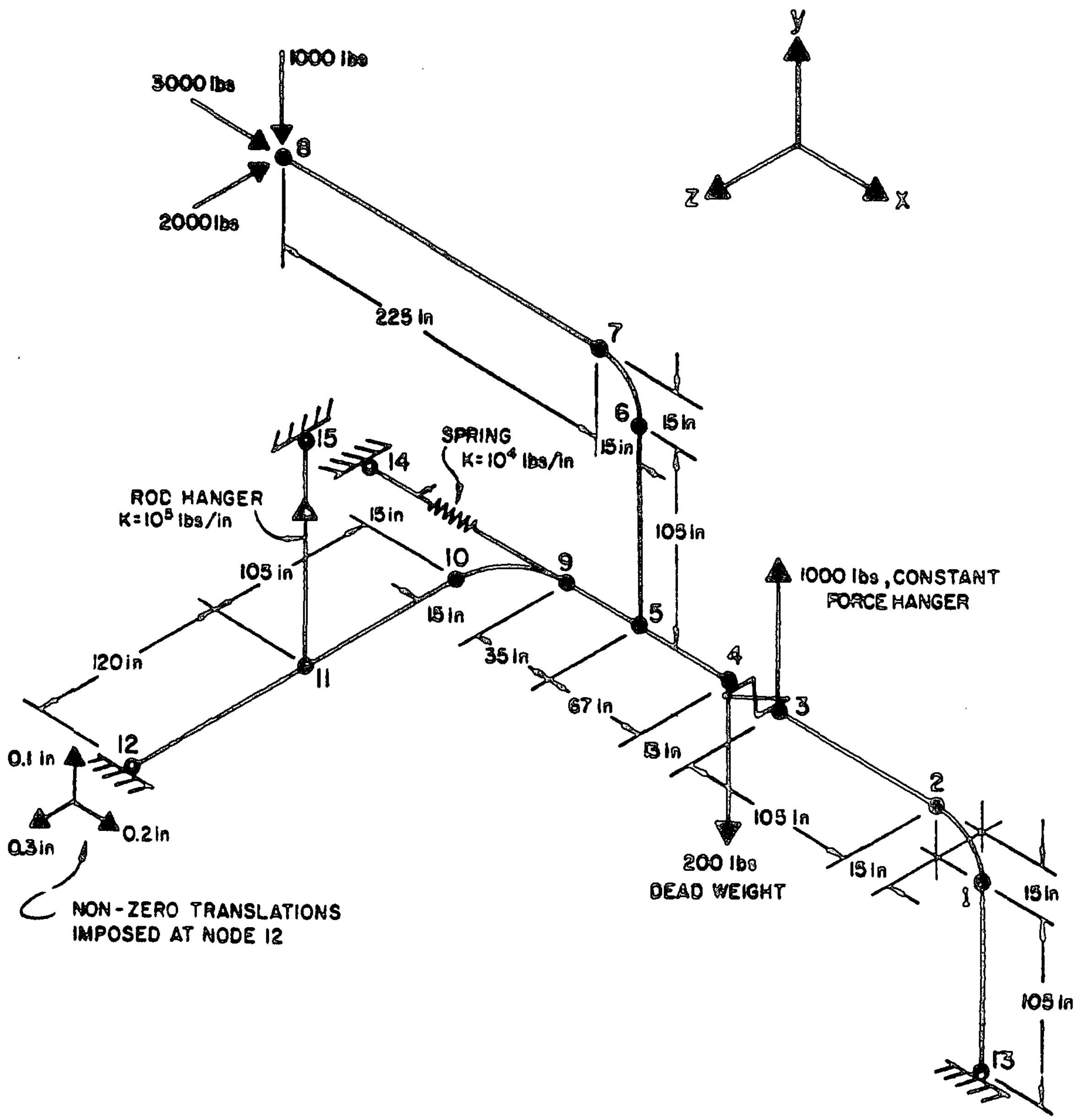
CIRCULAR PLATE ON A RIGID FOUNDATION
FOR SLSAP AND NOBEC



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-46

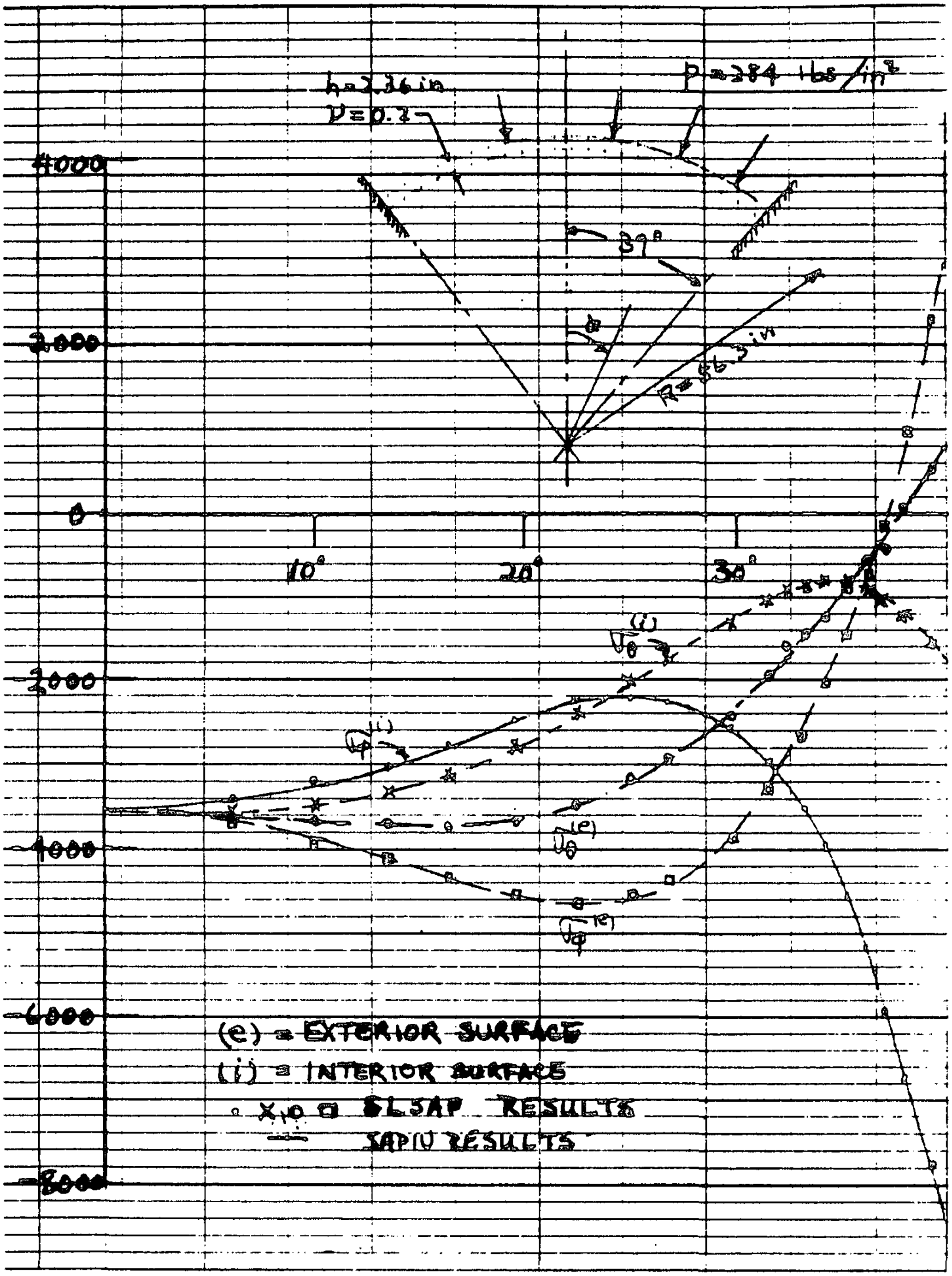
COMPARISON OF DISPLACEMENT AND MOMENT VARIATION OF CIRCULAR PLATE FROM SLSAP AND NOBEC



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

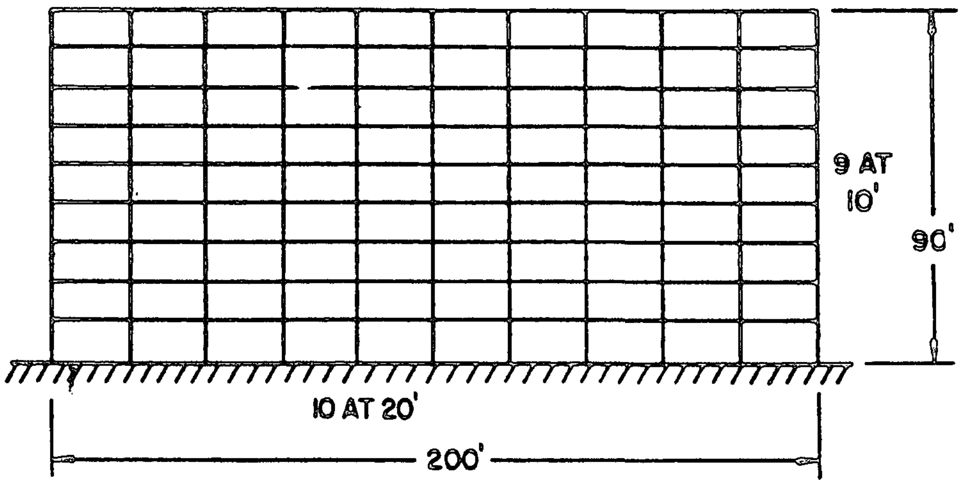
FIGURE D-47

MODEL OF PIPE NETWORK FOR SLSAP AND SAPIV
(SLSAP VALIDATION PROBLEM 1)



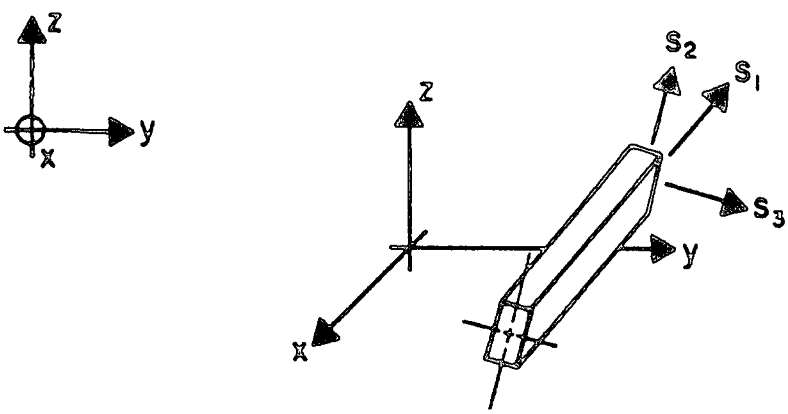
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-48
COMPARISON OF SURFACE STRESSES IN A
CLAMPED SPHERICAL SHELL UNDER
EXTERNAL PRESSURE FOR SLSAP AND SAPIV
(SLSAP VALIDATION PROBLEM 2)



(a) ELEVATION OF FRAME

DATA : YOUNG'S MODULUS = 432000 , MASS DENSITY = 1.0
FOR ALL BEAMS AND COLUMNS A_1 = 3.0, $I_1 = I_2 = I_3$ = 1.0
UNITS : FT, KIPS



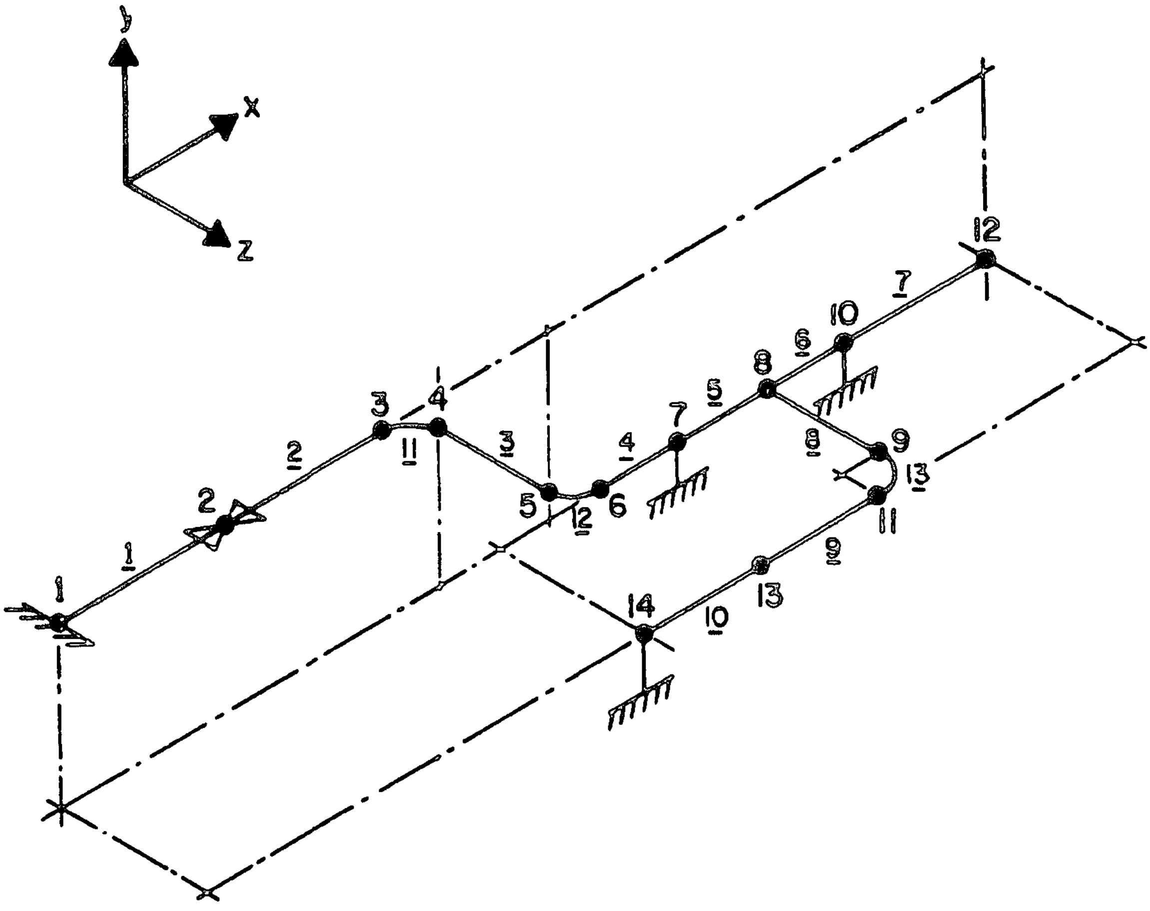
(b) BEAM ELEMENT DEFINITION

S_1, S_2 AND S_3 = BEAM LOCAL AXES

I_1, I_2 AND I_3 = FLEXURAL INERTIA ABOUT S_1, S_2, AND S_3

A_1 = AREA ASSOCIATED WITH S_1

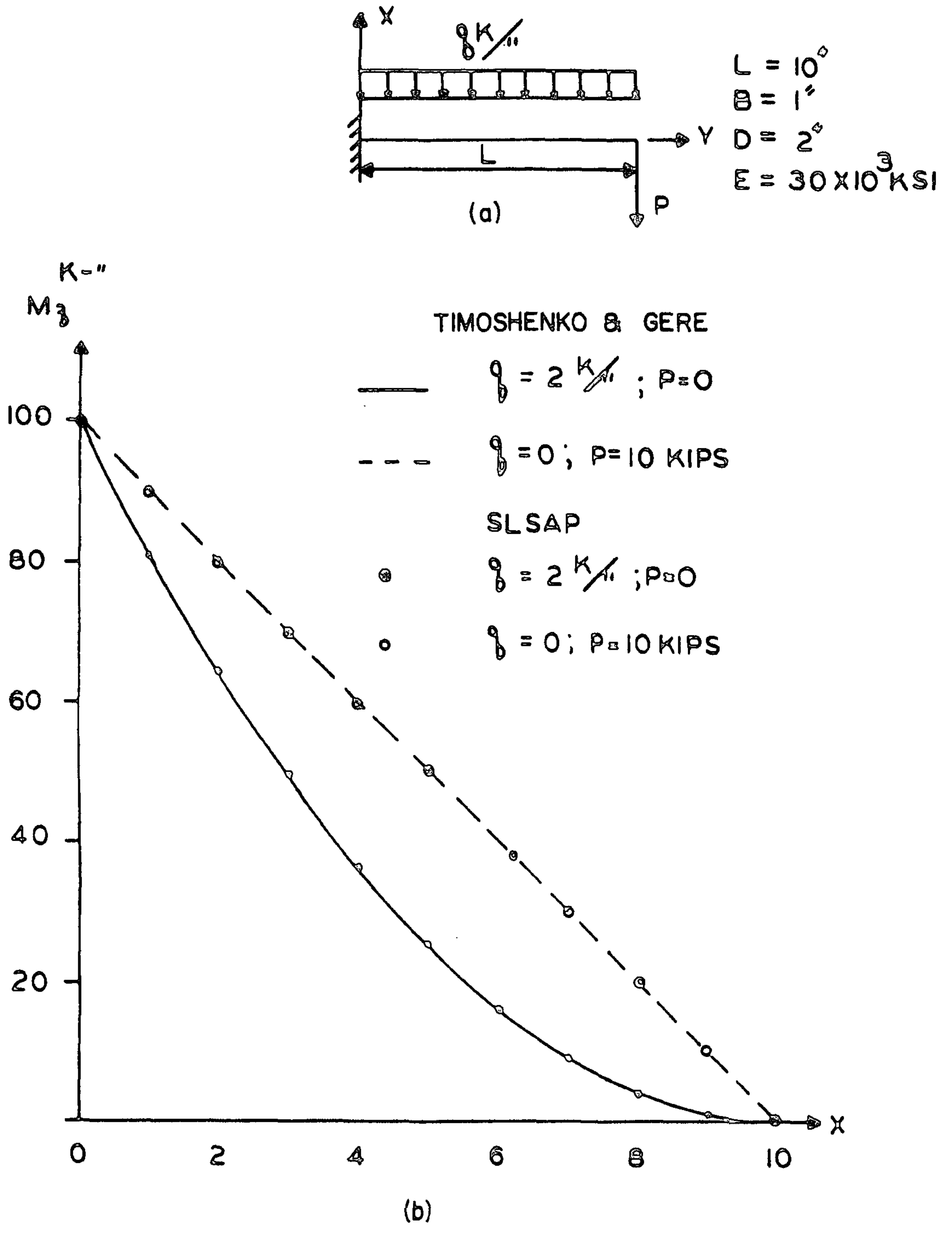
BYRON/BRAIDWOOD STATIONS UPDATED FINAL SAFETY ANALYSIS REPORT
FIGURE D-49 MODEL OF PLANE FRAME FOR SLSAP AND SAPIV (SLSAP VALIDATION PROBLEM 3)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-50

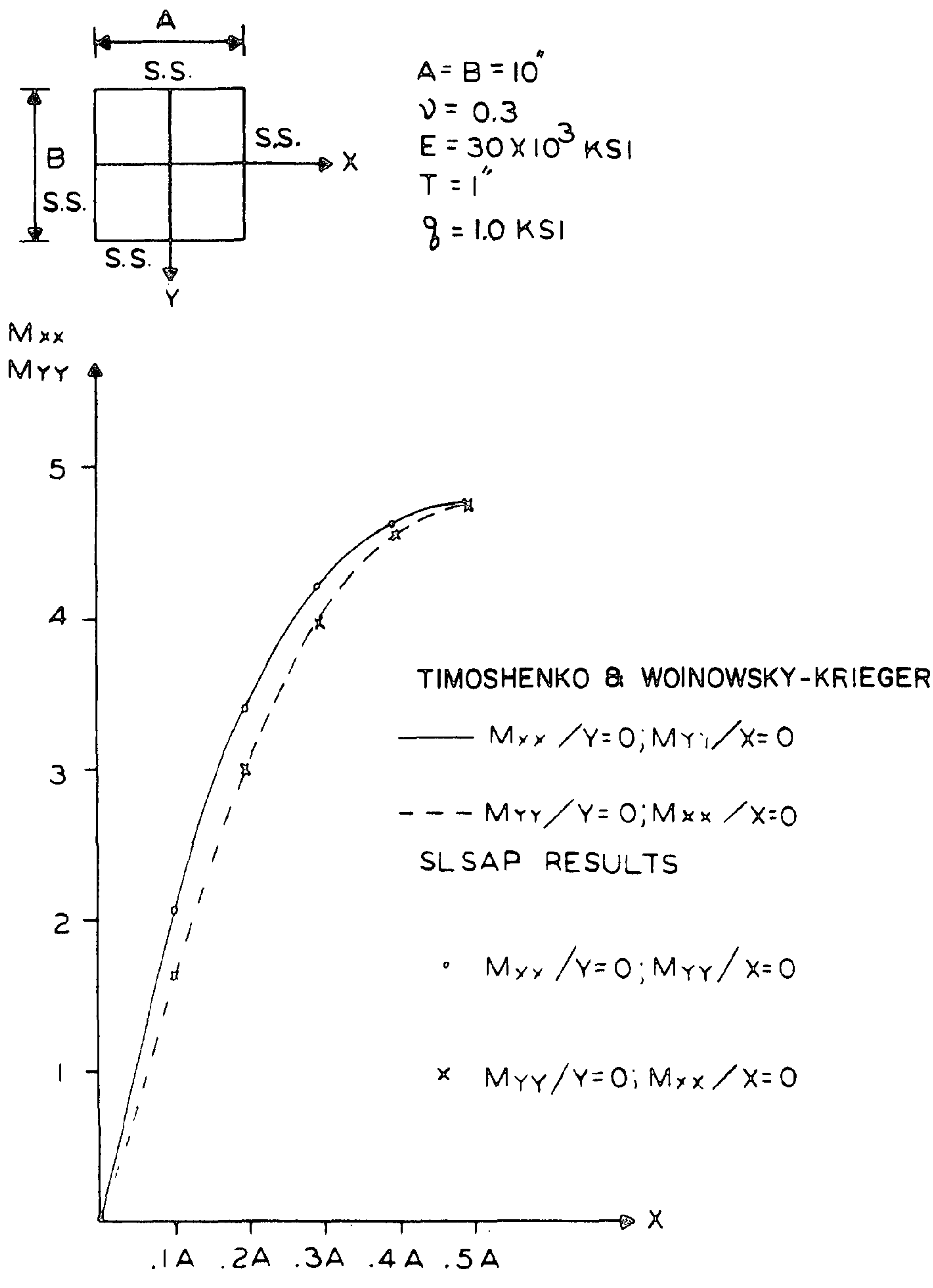
MODEL OF PIPE ASSEMBLAGE
FOR SLSAP AND SAPIV
(SLSAP VALIDATION PROBLEM 4)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-51

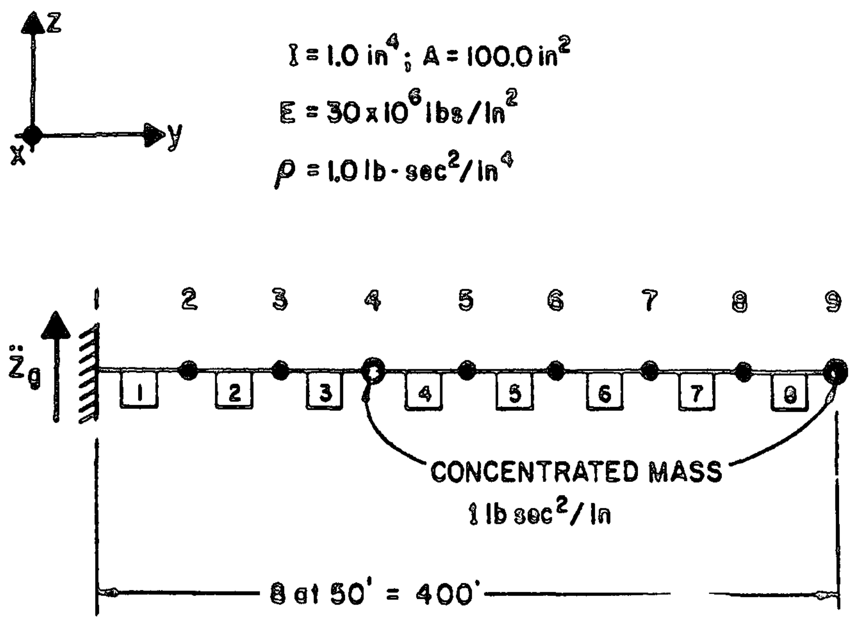
BENDING MOMENTS IN A CANTILEVER BEAM



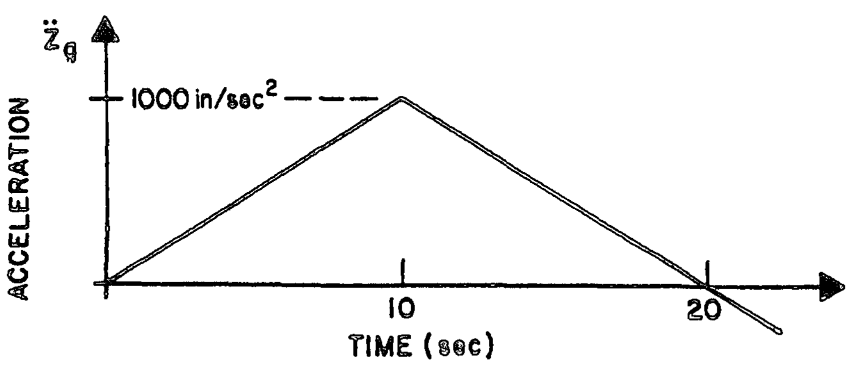
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-52

BENDING MOMENTS IN A SIMPLY SUPPORTED PLATE



(a) NODE AND BEAM NUMBER ASSIGNMENTS FOR THE CANTILEVER MODEL

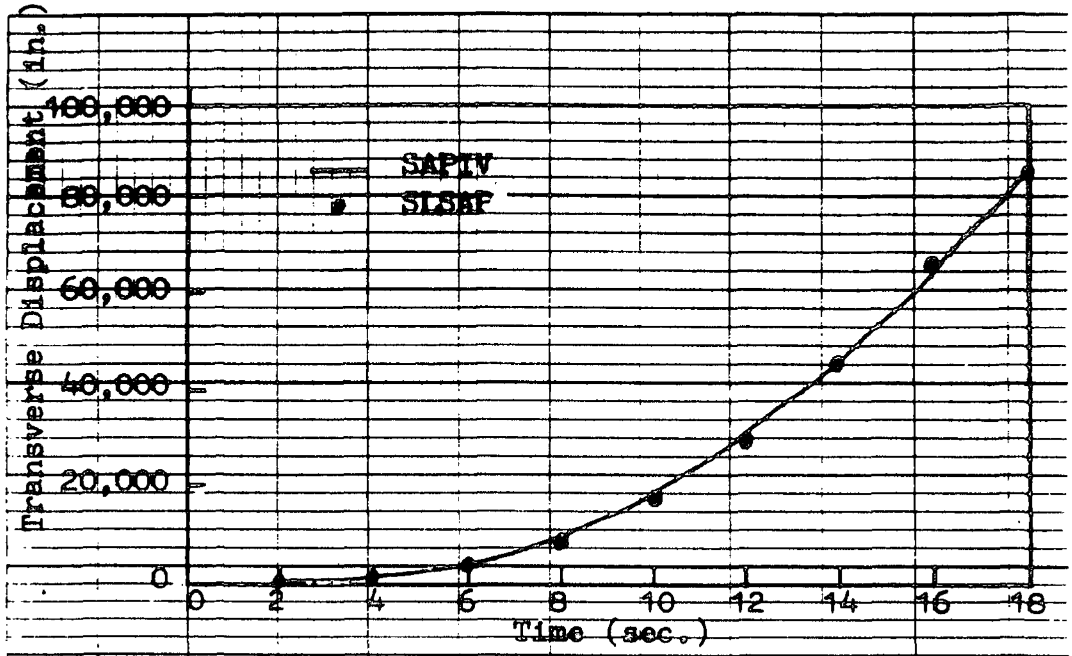


(b) GROUND ACCELERATION APPLIED AT NODE 1

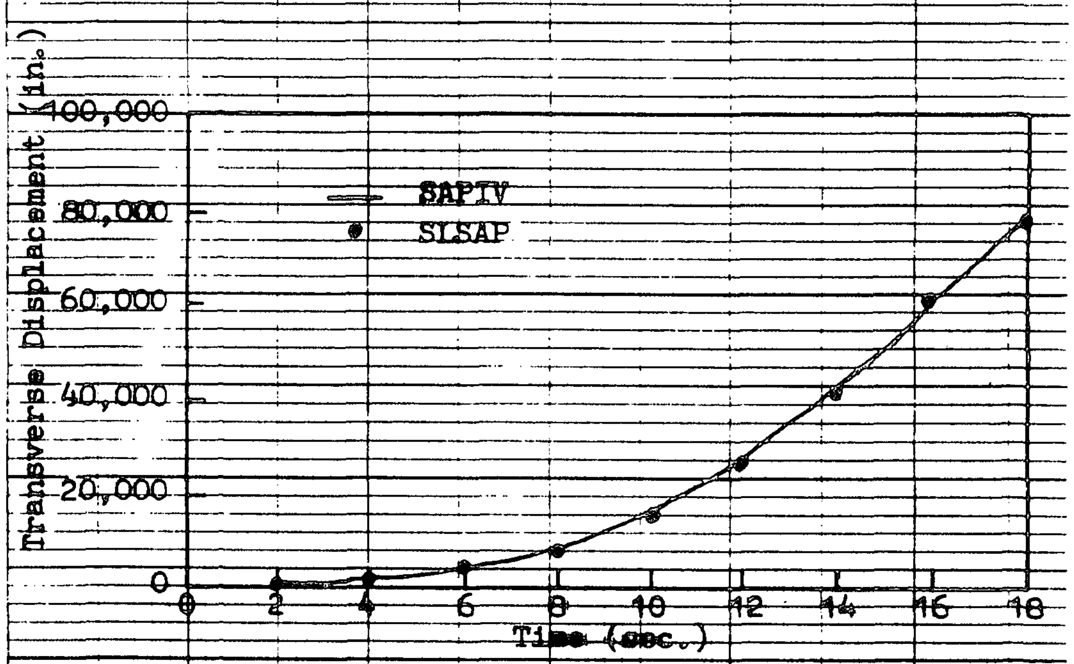
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-53

MODEL FOR RESPONSE HISTORY ANALYSIS FOR SLSAP AND SAPIV (SLSAP VALIDATION PROBLEM 7)



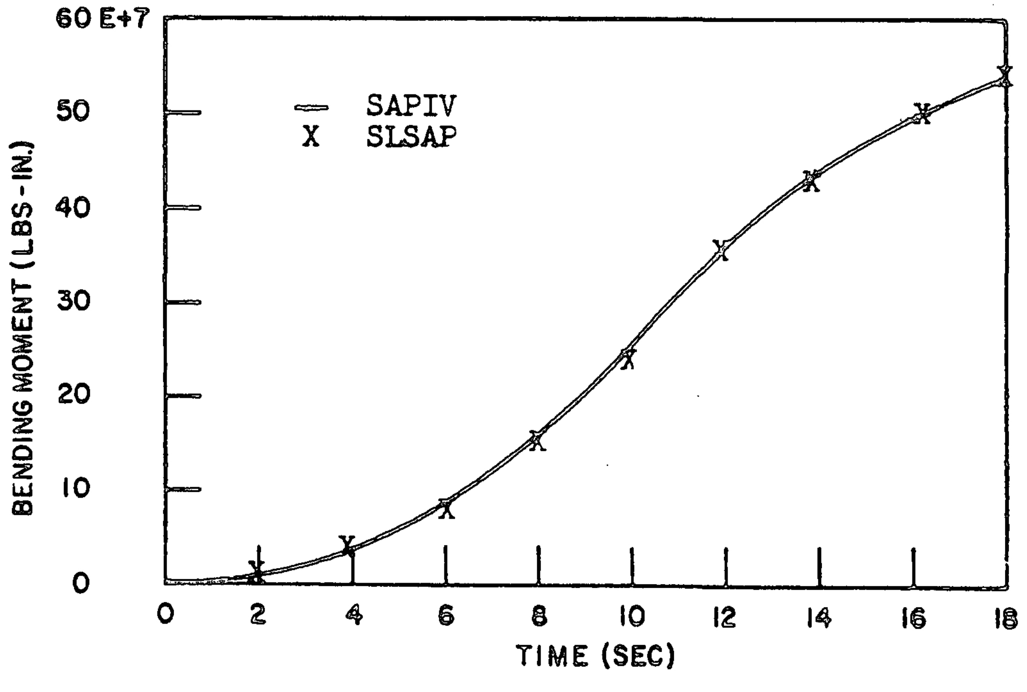
(a) Node 5



(b) Node 9

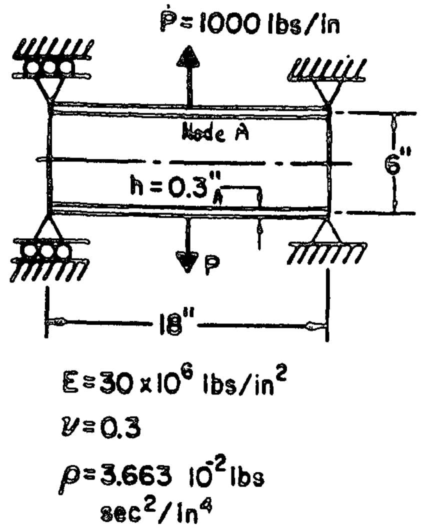
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-54
COMPARISON OF SLSAP AND SAPIV
TRANSVERSE DEFLECTIONS OF THE
CANTILEVER BEAM
(SLSAP VALIDATION PROBLEM 7)

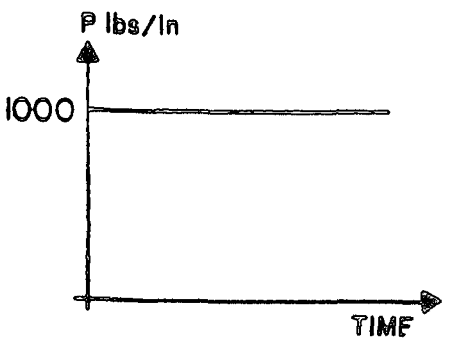


BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-55
COMPARISON OF SLSAP AND SAPIV
BENDING MOMENTS OF THE
CANTILEVER BEAM
(SLSAP VALIDATION PROBLEM 7)

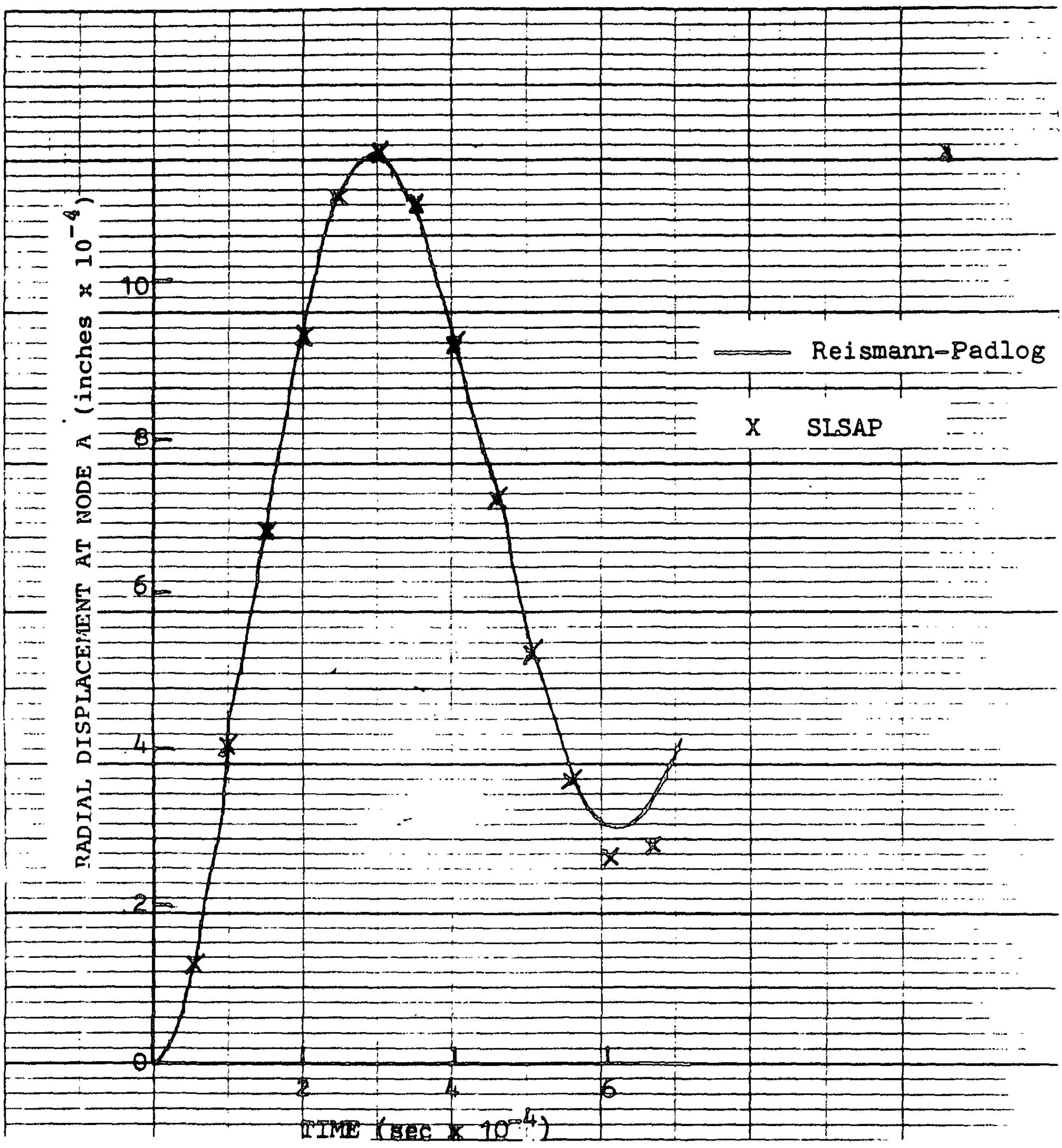


a) CYLINDRICAL TUBE



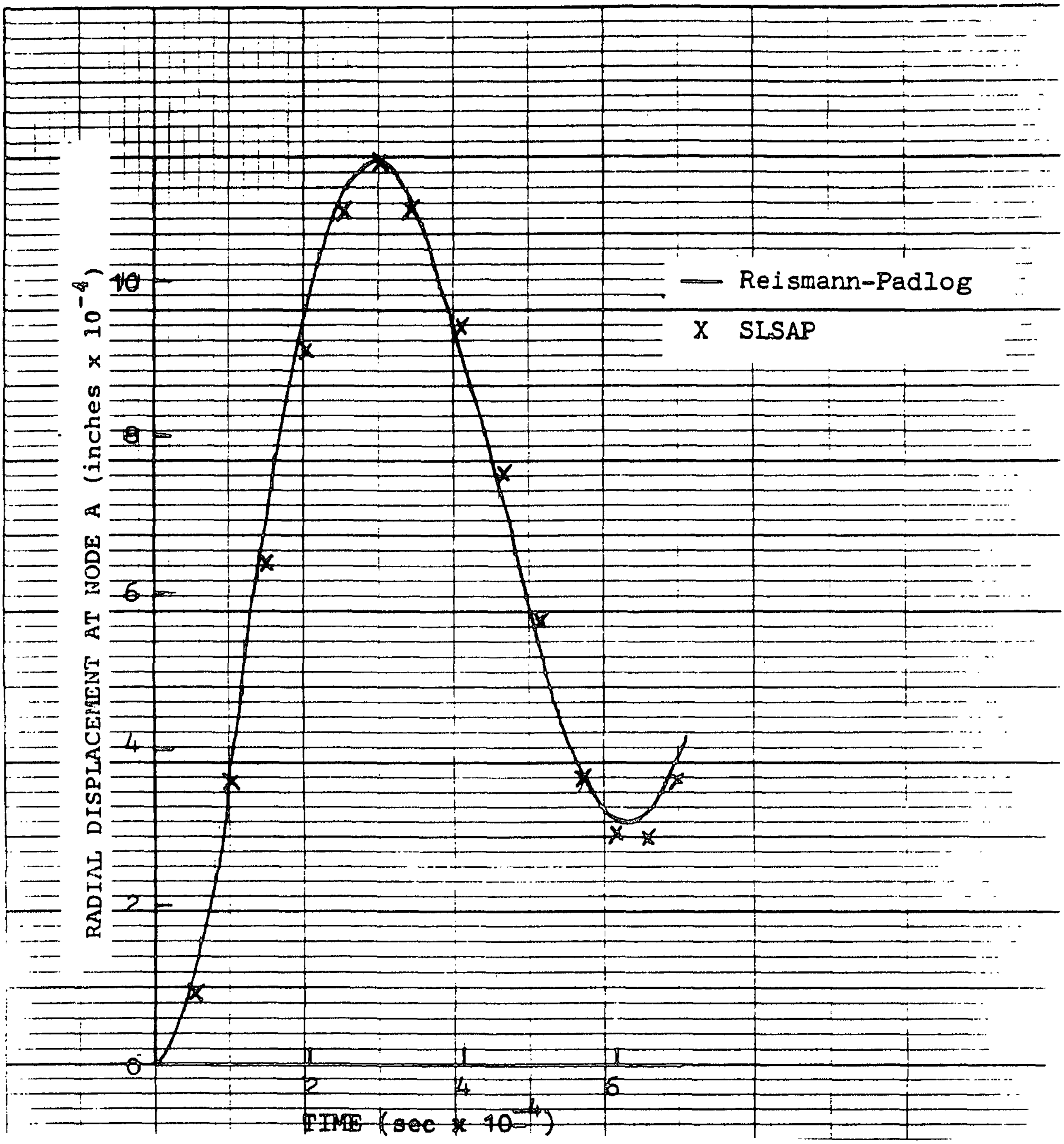
b) TIME VARIATION OF LOAD

BYRON/BRAIDWOOD STATIONS UPDATED FINAL SAFETY ANALYSIS REPORT
FIGURE D-56 CYLINDRICAL TUBE AND LOAD HISTORY FOR SLSAP AND SAPIV MODE SUPERPOSITION AND DIRECT INTEGRATION ANALYSES (SLSAP VALIDATION PROBLEM 8)



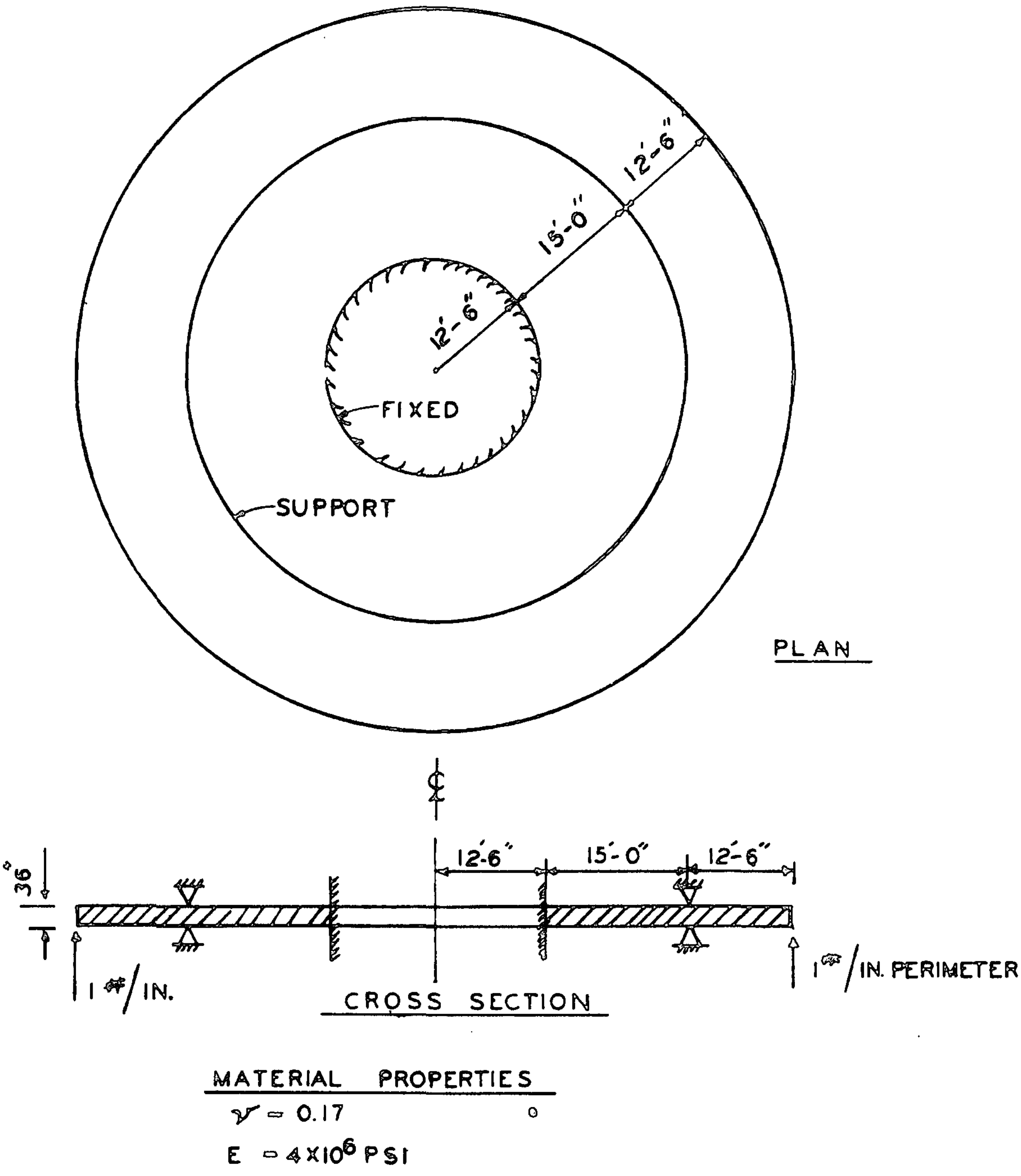
BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-57
DISPLACEMENT COMPARISON OF
SLSAP MODE SUPERPOSITION AND
REFERENCE 39 FOR THE CYLINDRICAL TUBE
(SLSAP VALIDATION PROBLEM 8)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

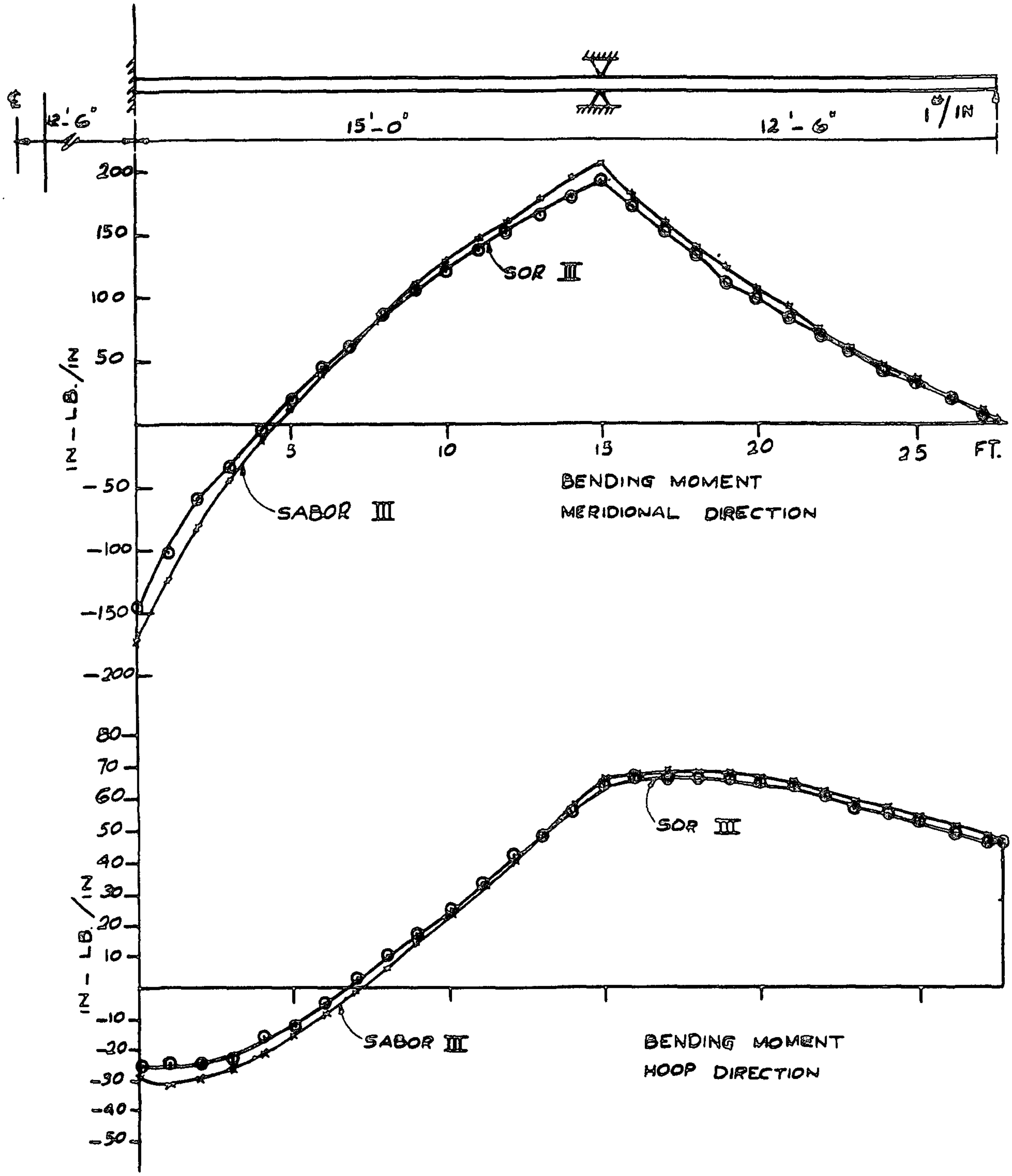
FIGURE D-58
DISPLACEMENT COMPARISON OF
SLSAP DIRECT INTEGRATION AND
REFERENCE 39 FOR THE CYLINDRICAL TUBE
(SLSAP VALIDATION PROBLEM 8)



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-59

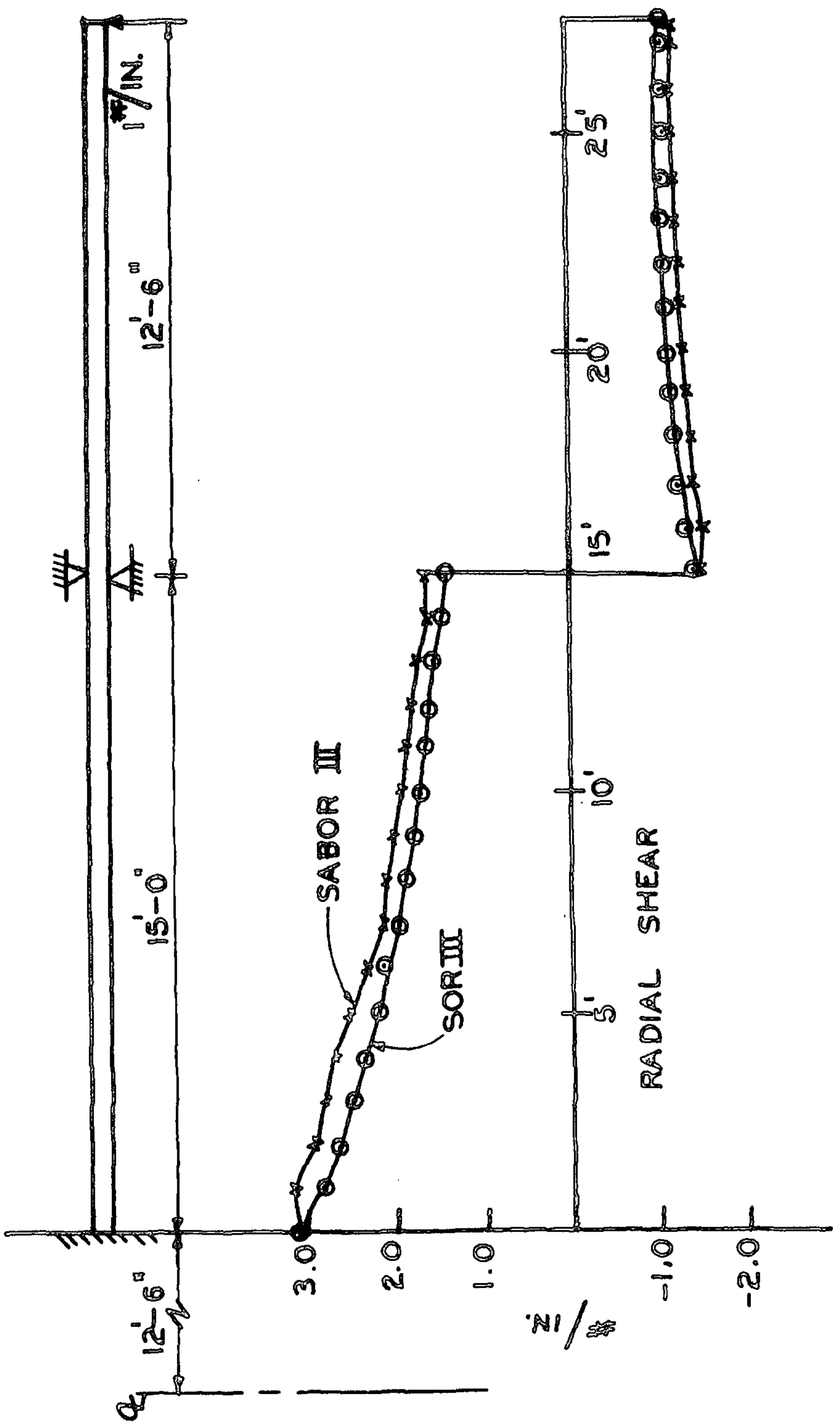
CIRCULAR PLATE FOR SOR-III EXAMPLE



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-60

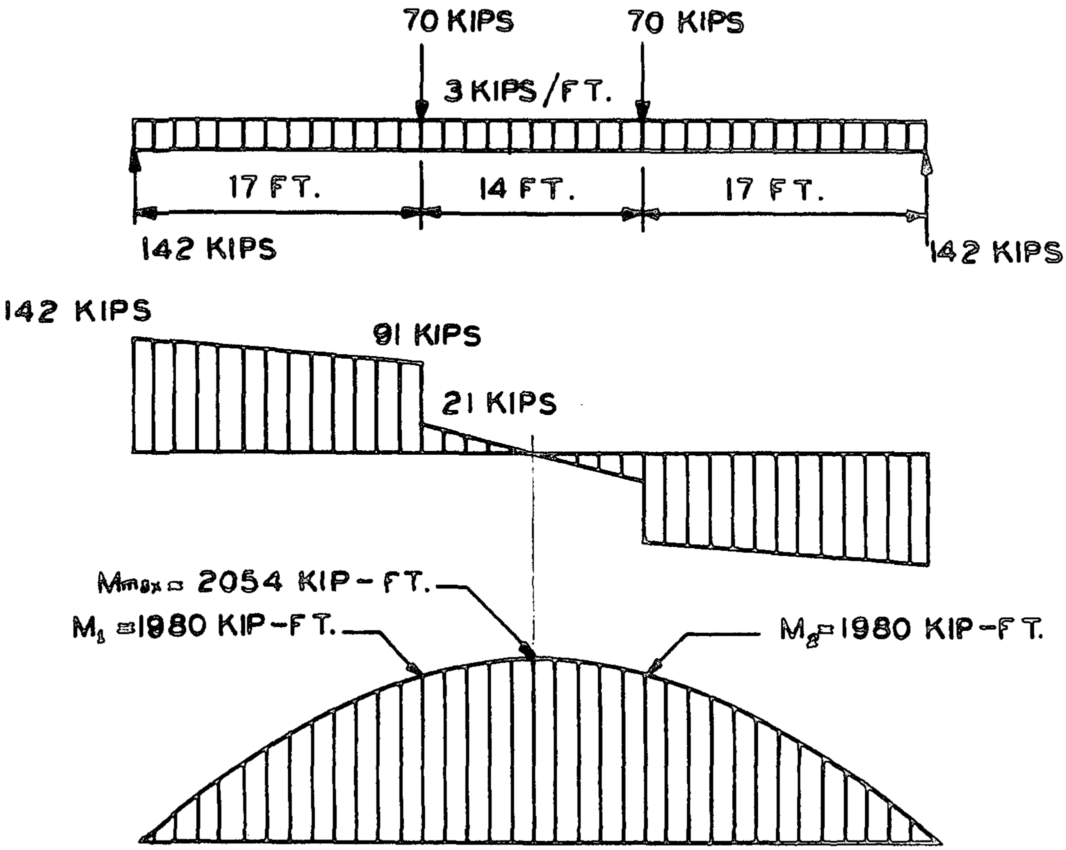
MOMENT COMPARISON OF SABOR-III AND SOR-III



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-61

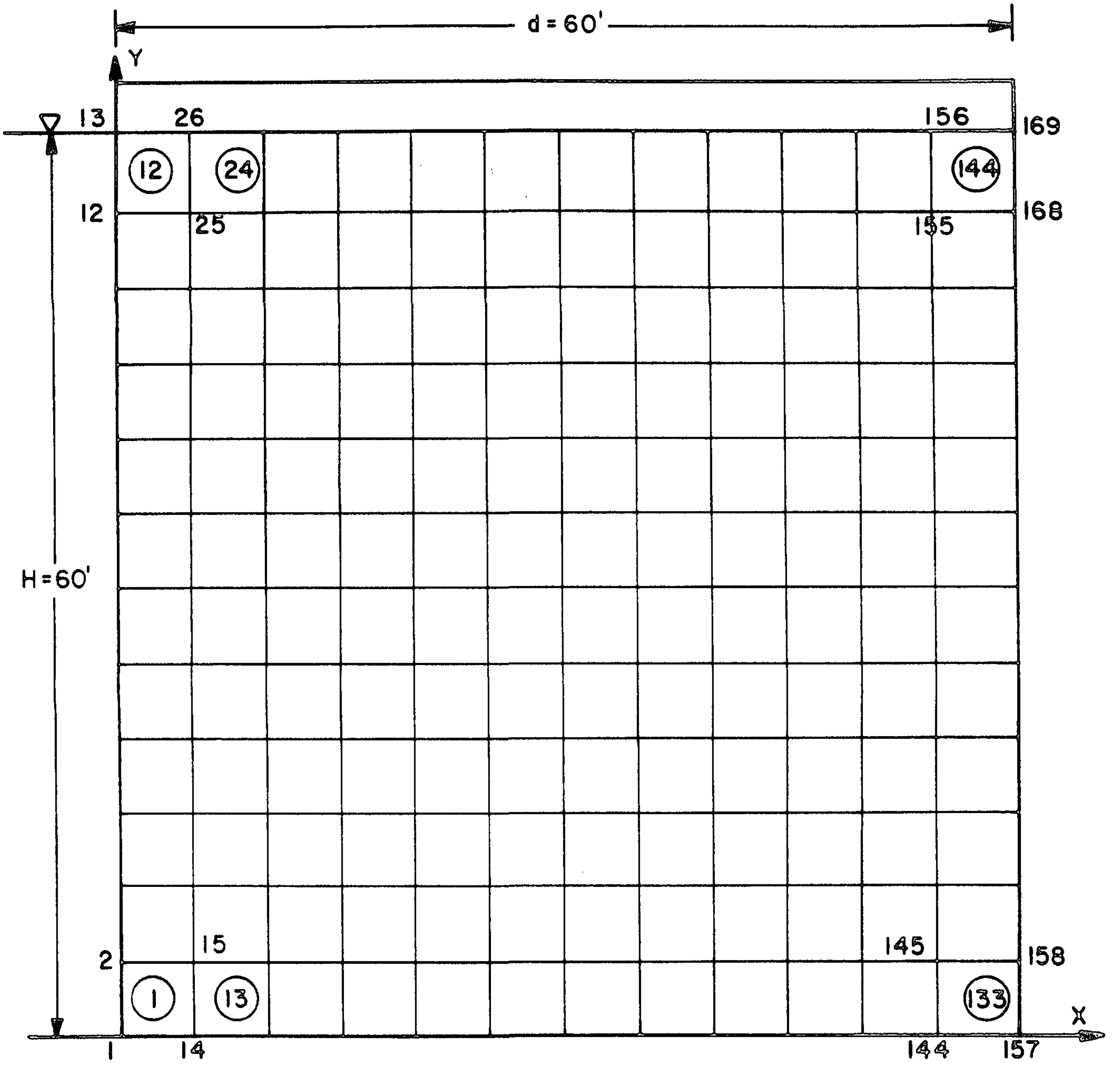
RADIAL SHEAR COMPARISON FOR SABOR-III AND SOR-III



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-62

SHEAR AND MOMENT DIAGRAMS

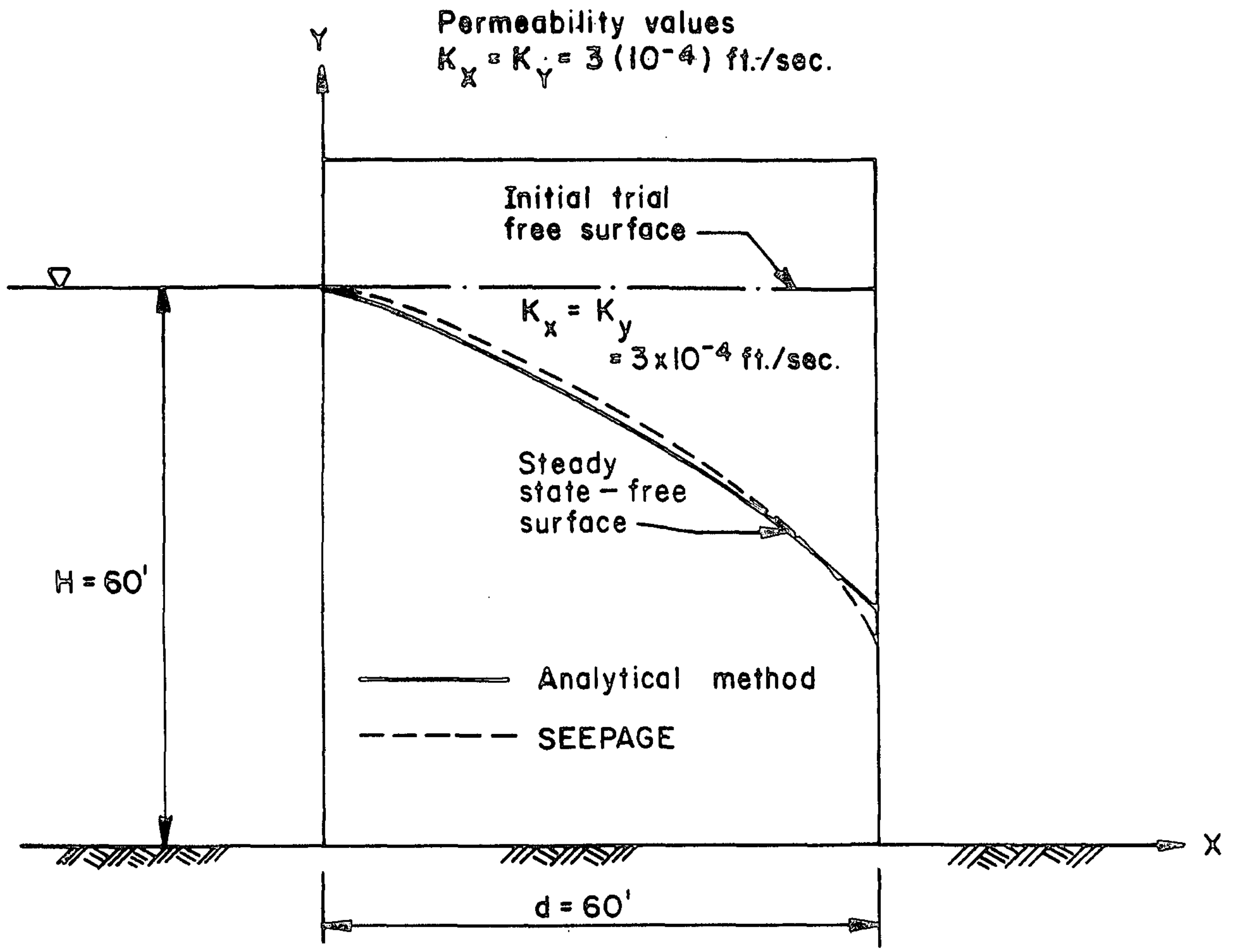


Numbers indicate nodal identification
Circled numbers indicate element identification

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-63

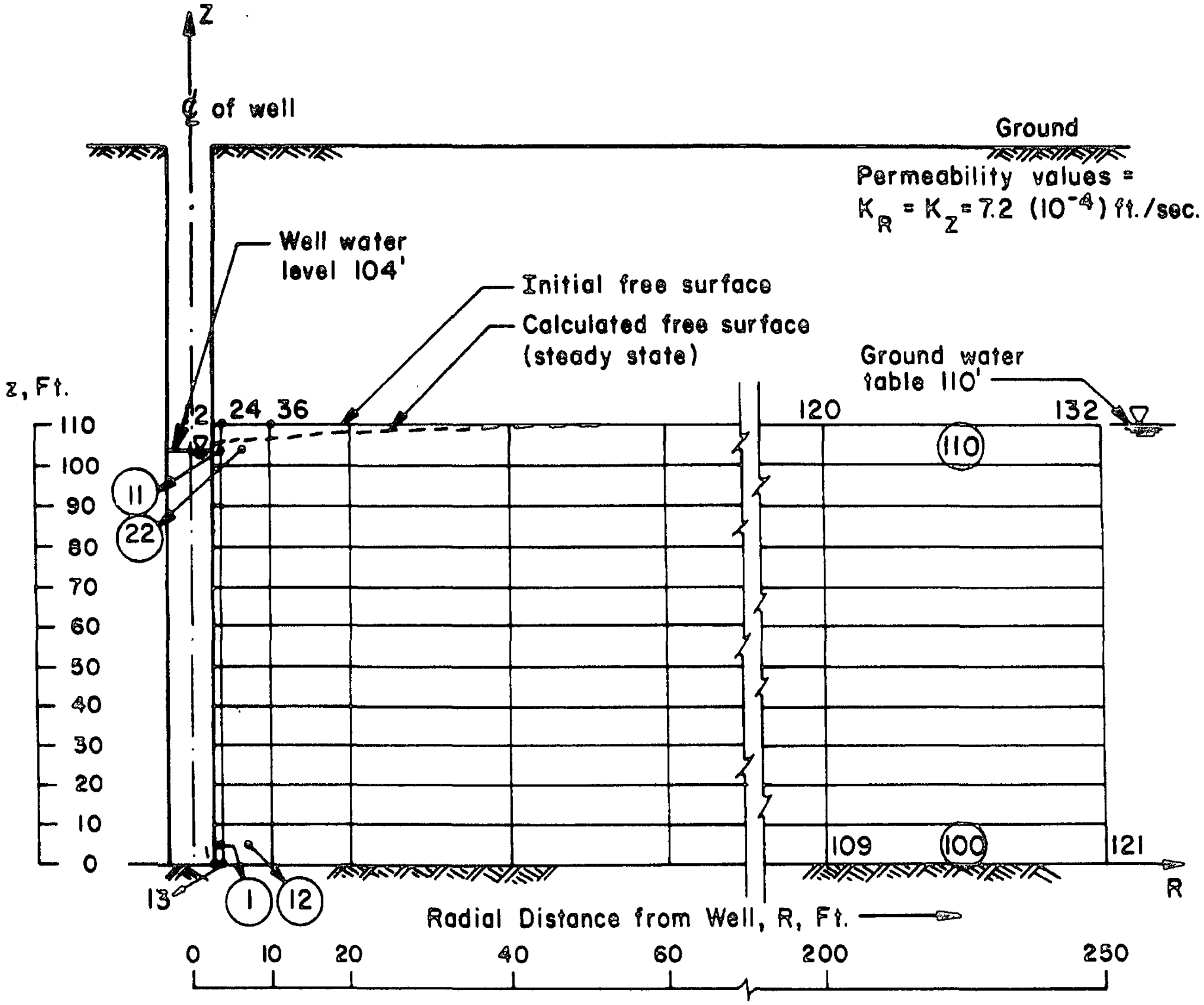
PLANE FLOW PROBLEM AND THE
FINITE ELEMENT MESH



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-64

COMPARISON OF FREE SURFACES OBTAINED FROM SEEPAGE AND ANALYTICAL METHOD

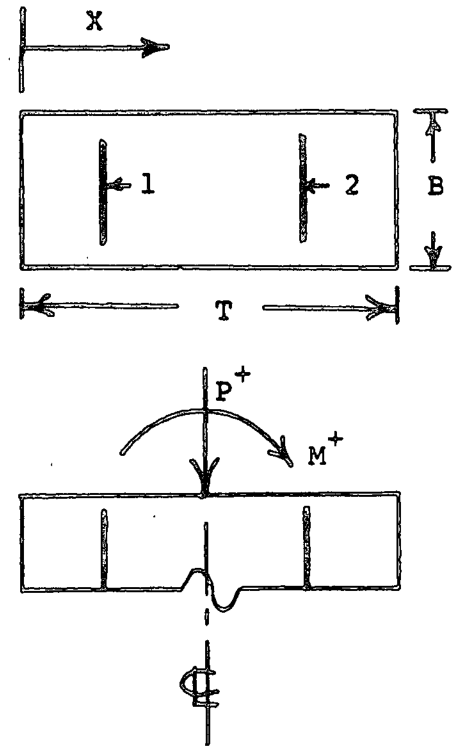


Numbers indicate the nodal identification
Circled numbers indicate element identification

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-65

FINITE ELEMENT MESH FOR
AXISYMMETRIC FLOW PROBLEM



CHARACTERISTICS

B = Width of Section

T = Thickness of Section

X_1 = Location of Reinforcing Layer 1

A_{s1} = Area of Steel in Layer 1

X_2 = Location of Reinforcing Layer 2

A_{s2} = Area of Steel in Layer 2

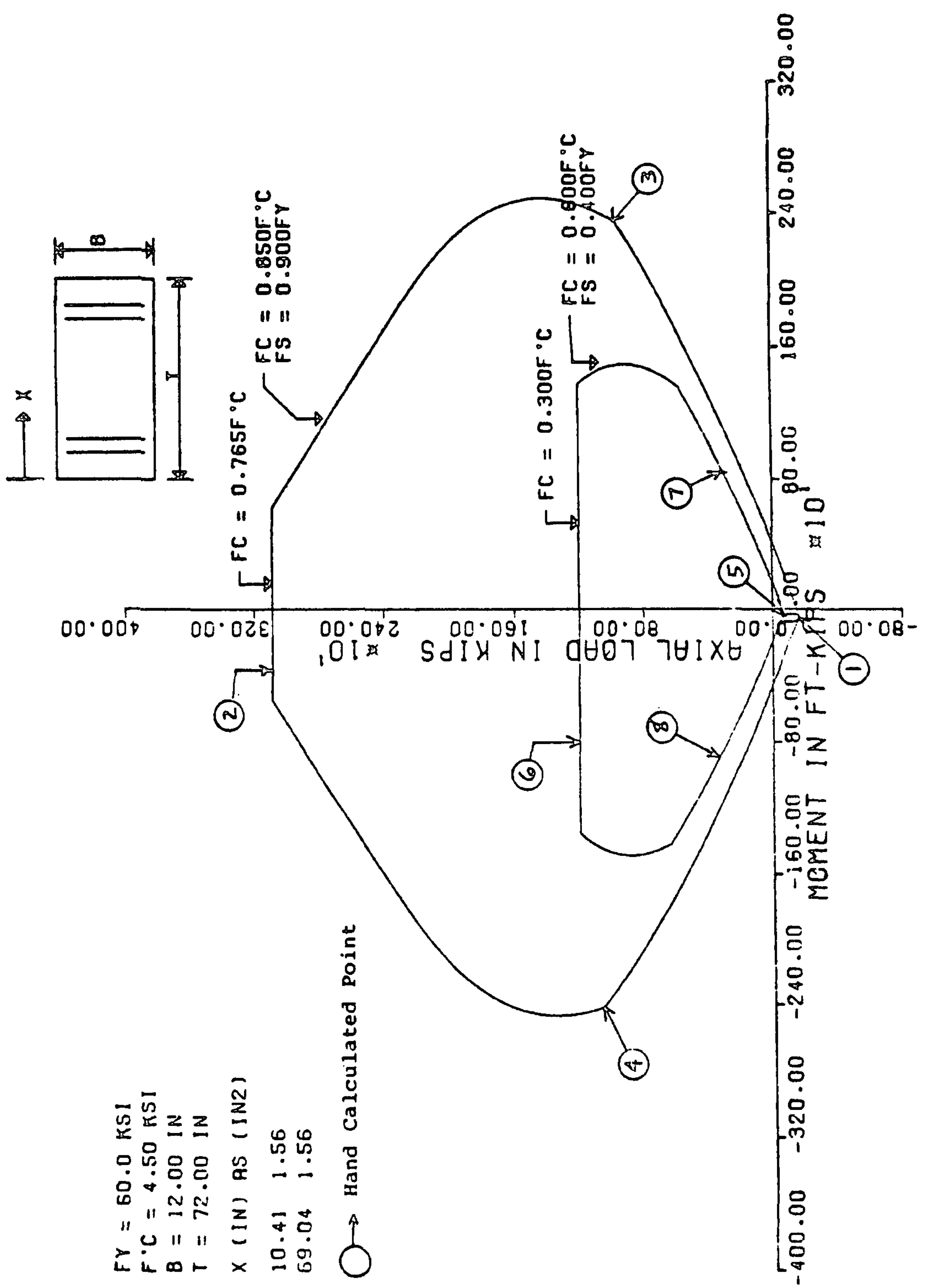
F_y = Steel Yield Strength

F'_c = Concrete Strength

BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-66

COLID RECTANGULAR SECTION PARAMETERS
AND SIGN CONVENTION



BYRON/BRAIDWOOD STATIONS
UPDATED FINAL SAFETY ANALYSIS REPORT

FIGURE D-67

COLID INTERACTION DIAGRAM FOR RECTANGULAR SECTION STRESS FACTOR PROBLEM